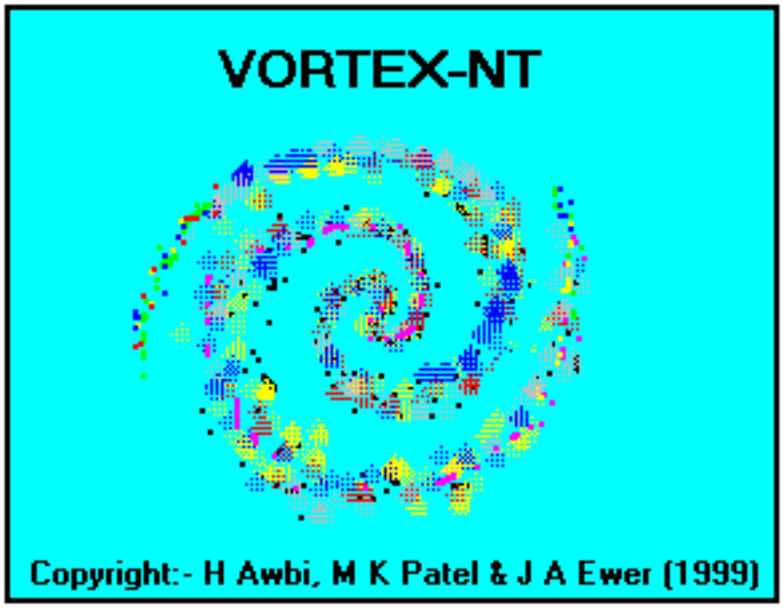
VORTEX©-NT USER MANUAL

FOR AIR FLOW, HEAT TRANSFER AND CONCENTRATION IN ENCLOSURES

VERSION 3.10-005W
December 2001



VORTEX© *is the copyright of Dr. H. B. Awbi*
FLOWVIS© *is the copyright of J.A. Ewer and Dr. M.K. Patel*

Contents *Page*

1. INTRODUCTION 5

2. REQUIREMENTS FOR USING VORTEX 6

3. INSTALLATION AND RUNNING 7

4. DESCRIPTION OF THE CFD MODEL 8

5. MODELLING TECHNIQUES 9

INPUT PARAMETERS 9
FIRE SOURCE DATA 9
CREATING *.VDB FILE 9

6. RUNNING VORTEX 12

RUN PAGE 12

OUTPUT FILES 12

7. OTHER INPUT DATA FILES 14

FILE MODEL.DAT 14
FILE FABRIC.DAT 14

8. EXAMPLES 17

EXAMPLE 1 17
EXAMPLE 2 17
EXAMPLE 3 18

9. INTRODUCTION TO FLOWVIS 46

10. HARDWARE AND SOFTWARE REQUIREMENTS 47

11. OVERVIEW OF *FLOWVIS* 48

12. GETTING STARTED WITH *FLOWVIS* 50

13. MAIN MENU OPTIONS 51

GRAPH 53
STREAMLINES 56
GEOMETRY 58
GRID 62
VECTORS 63
CONTOUR FILLS 68
HELP 70
FILE MANAGER 71
DISPLAY SELECTION 77
TEXT 79
REDRAW 80
STACK 81
DELETE 82
PRINT 83
SETTINGS MENU 85
SYSTEM 87
EXPERT 88

LIMITATIONS OF *FLOWVIS* (VERSION 3.10) 89

KNOWN BUGS IN *FLOWVIS* (VERSION 3.10) 90

***FLOWVIS* FACILITIES (VERSION 3.10) 91**

PLANNED EXTENSIONS TO *FLOWVIS* 94

NOMENCLATURE		UNITS
C	mean concentration	(ppm or mg/m^3)
C_p	specific heat of air at constant pressure	(J/kgK)
C_1,C_2,C_D,C_μ	constants of turbulence model	
CIN	concentration of inlet air	(ppm or mg/m^3)
DELTA_	thickness of building fabric	(m)
E	east wall	
EPSO_	emissivity of room surface	
G	energy generation term	
g	gravitational acceleration	(m/s^2)
H	room height in y direction	(m)
HC_(I)	convective heat flux for room surface I	(W/m^2)
HOBS(I)	heat flux from obstacle I	(W/m^2)
HR_(I)	radiant heat transfer flux for room surface I	(W/m^2)
HWIN_(I)	heat flux for window I	(W/m^2)
h	convective heat transfer coefficient (W/m^2K)	
I, J, K	grid point number in x, y, z direction	
ICLO	thermal resistance of clothing (m^2K/W)	
ICOMF	flag for thermal comfort calculations	(1 or 0)
ICON_(I)	grid coordinate of concentration source I	
ICONC	flag for concentration calculations	(1 or 0)
ICOND	flag for fabric heat conduction calculation	(1 or 0)
IDR	flag for draught risk calculation	(1 or 0)
IRAD	flag for surface radiation calculation	(1 or 0)
IWIN_(I)	grid coordinate for window I	
k, TK_	thermal conductivity	(W/mK)
L	room length in x direction, or left wall	(m)
LAYER_	building fabric layer	
M	metabolic rate	(W/m^2)
MAXIT	maximum number of iterations	
N	north wall	
NCON	number of concentration sources	
NI, NJ, NK	total number of grid points in x, y, z directions	
NIN	number of inlets	
NP	number of monitoring points	
NOBS	number of obstacles	
NOUT	number of outlets	
NWIN	number of windows	
PMV	predicted mean vote	
PPD	predicted percentage of dissatisfied	(%)
PA	water vapour pressure	(Pa)
p	pressure	(Pa)
q	heat production	(W/m^3)
R	right wall	
S	south wall	
S_ϕ	source term	
t	time	(s)
T_	inside room surface temperature	(^oC)
TIN	inlet air temperature	(^oC)
TAO_	outside air temperature	(^oC)

TMRT	local mean radiant temperature	(^oC)
TOBS(I)	temperature of obstacle I	(^oC)
TURBIN	turbulence intensity of air at supply outlet	
U_f	floor heat conduction coefficient (W/m^2K)	
U_i	tensor velocity component	(m/s)
U_t	heat transmission coefficient for a bare floor (W/m^2K)	
U,V,W	mean velocity components in x, y, z directions	(m/s)
VIN	inlet velocity	(m/s)
VAO_	outdoor air velocity	(m/s)
W	room width in z direction, or west wall	(m)
WIN	window coordinates	
X(I)	position of grid I in x direction	(m)
x_n	thickness of layer n	(m)
Y(J)	position of grid J in y direction	(m)
Z(K)	position of grid K in z direction	(m)

Greek letters

ß	volumetric expansion coefficient	(1/K)
ε	turbulence dissipation rate	
ϕ	dependent variable	
κ	Karman's constant (= 0.4187)	
Γ_ϕ	diffusion coefficient	
μ, μ_t	laminar and turbulent viscosities of air	(Pa s)
ρ	air density	(kg/m^3)
σ_l, σ_t	laminar and turbulent Prandtl numbers	

Subscripts

B	buoyancy
c	concentration
e	effective
i	vector dimension
k	kinetic energy
s	shear
t	turbulent
ε	kinetic energy dissipation

1. INTRODUCTION

Air distribution systems in modern buildings require careful design to meet the increasingly stringent demands for thermal comfort, indoor air quality and energy efficiency.

Although procedures are available for designing conventional air distribution systems, designers have to rely on data obtained from a physical model of the proposed air distribution scheme when dealing with a non-conventional system. Modifications to the model are then made until the desired conditions are achieved. This procedure is very costly and time consuming. In addition, it is not always possible to construct full scale physical models. Under such circumstances, computer based analysis would be extremely useful in the preliminary and actual design stages.

Recently, there has been much activity in the field of Computational Fluid Dynamics (CFD) to develop computer codes which numerically solve the differential equations that govern air movement, heat transfer and the distribution of chemical species for internal and external flows. Today, there are few codes which are dedicated to the built environment.

VORTEX© (**V**entilation **O**f **R**ooms with **T**urbulence and **E**nergy e**X**change) is a program which has been specially developed to simulate the air movement, heat and concentration of species transport in buildings. The software gives all the information that can be obtained from physical model tests, and more, at a very low cost and within minutes or few hours, depending on the complexity of the problem. It can also produce a full thermal comfort assessment of the indoor environment using the ISO Standard 7730 method. The program has been under development for over eight years by specialists in ventilation and room air movement research. A pre-processing environment has recently been added to automate the data facility through the use of "windows-look-alike" environment. The visualisation of the results is performed using a post-processing tool called **FLOWVIS**©.

2. REQUIREMENTS FOR USING VORTEX

VORTEX is a CFD code, which has been specifically developed for airflow, heat transfer and concentration of species predictions in buildings. Unlike other general-purpose CFD codes, scientists and engineers who have basic computing skills but not necessarily CFD experience can use VORTEX. This manual contains examples that can be accessed by the user to gain familiarity with the type of data required to perform a CFD simulation. However, a thorough understanding of the problem being solved is essential to produce an "electronic model" of the building before a reliable solution can be achieved.

VORTEX© is available for a workstation using **UNIX,** or for a PC using **DOS 5** or higher. A minimum of 8 Mb. RAM of available memory is required to solve small or medium size problems. For larger problems involving extensive number of grid points, 16 Mb. or 32 Mb. RAM may be required. An Intel 486 DX processor or a Pentium will be expected to ensure acceptable CPU run time

3. INSTALLATION AND RUNNING

INSTALLING VORTEX.

The software can be installed by running the SETUP_310-005W.EXE program from the media provided.

NOTE: Install the software on the root drive on your machine, (e.g. D:\ because the installation program will automatically generate the necessary directories that are needed.

The machine also needs to be rebooted.

The software can be run using the VORTEX_310-005W icon. This will automatically run in the directory **DRIVE:\VORTEX\V310\VORT_WNT\WORKAREA**

THE LOCKING DEVICE

The locking method used by VORTEX is one that generates a unique string that recognizes your machine. You require no hardware keys, but only strings, however the user has that in order to be provided with a unlocking string the following steps need to followed:

After installing the software on the hard disk run **VORTEX using the model file SEND3.** This generates a file called **SENDFILE.USR** on the C:\ drive.

Send the file **SENDFILE.USR** to the distributor either via email or fax.

On receiving an unlocking string, file name called **NTUSER31.INF**, please copy this file in the directory **DRIVE:\VORTEX\V310\VORT_WNT\BIN.**

if you have any problems running the software, please note any error numbers and/or messages and contact the distributor stating these.

4. DESCRIPTION OF THE CFD MODEL

The general transport equations are the continuity equation, the Navier-Stokes (momentum) equation, the thermal energy equation, the concentration of species equation and the two equations for the kinetic energy and dissipation rate of the k-ε turbulence model. For incompressible flow, the time-averaged equations are represented by:

$$\underbrace{\frac{\partial(\rho\phi)}{\partial t}}_{\text{Transient}} + \underbrace{\frac{\partial}{\partial x_i}(\rho U_i \phi}_{\text{Convection}} \underbrace{- \Gamma_\phi \frac{\partial \phi}{\partial x_i})}_{\text{Diffusion}} = \underbrace{S_\phi 0}_{\text{Source}} \qquad (1)$$

where S_ϕ is the source term for the variable ϕ (see Table 1).

To obtain a solution of equation (1) the boundary conditions for the room or building must be specified. These are known quantities, empirical or semi-empirical expressions. Typical room boundary conditions, which must be provided, are the velocity, temperature and the contaminant concentration of the supply air. The air velocity, temperature and contaminant concentration at the exit are obtained from the continuity equation, thermal energy and concentration mass balance equations respectively. For incompressible flow as in buildings, the air density is unaffected by pressure and therefore the pressure is a relative quantity and only pressure difference is a significant entity.

At a solid surface a no-slip condition is imposed, i.e. zero velocity. The wall surface temperature are either specified or computed from the heat transfer rate when a heat flux is specified at the surface.

The airflow equations are discretized and solved in a staggered 3-D Cartesian system using the Finite Volume Method (FVM) and the well-known SIMPLE algorithm. The Hybrid Scheme (i.e. a combination of Central Difference and Upwind Schemes) is used as the finite difference schemes for solving the equations. To enhance the stability of the solution process under-relaxation techniques are applied to all the equations.

5. MODELLING TECHNIQUES

Input Parameters

To perform a flow simulation-using **VORTEX** one has to generate a problem specific data file called **model.VDB**. The pre-processor generates this file after the user specifies the required variables. In general, the user specifies the internal room dimensions, the inlet and outlet openings, the obstructions in the room, the position of heat or contamination sources, etc. The pre-processor to suit the room configuration then generates a computational grid. The user has the option to change the grid setting if so desired. The heat sources at the room surfaces and those produced by certain obstructions are specified in W/m^2 of surface area. The user also specifies the inlet air velocity, turbulence intensity, temperature and concentration.

When all the data is specified and clicking on the File icon saves the model.VDB file for the case, a **model.DAT** file is automatically generated by the pre-processor and stored in the directory **DRIVE:\VORTEX\V310\VORT_NT\WORKAREA**. The parameters, which are included in the model.DAT file, are given in Table 2 and in the Examples at the end of this manual. See Section 7 for mode details.

Fire Source Data

For simulating a fire source in a room, the source is specified as a thin obstacle with a surface temperature (**TOBS(I)**) $\geq 500.0^{o}C$ and a heat flux (**HOBS(I)**) in kW/m^2. The heat flux increases with time up to a time lapse of 180 seconds using the following:

$$Q_{t+dt} = 2^{dt/60}\ Q_t$$

where Q_{t+dt} is the heat flux at time t+dt seconds in kW/m^2
Q_t is the heat flux at time t seconds in kW/m^2 ; and
dt is the time step in seconds.

The heat flux remains constant after 180 seconds.

*Creating *.VDB File*

File Page
To create a new **model.VDB** file or editing an existing one, the **File** icon of the Vortex Setup Tool must be opened. For a new case, the name of the case is specified in the **Current File** box using 8 characters maximum (without the **VDB** extension). The file is then saved before the **Config** page is opened. For loading an existing **model.VDB** file, the **Load** box is clicked.

Config Page
In the Config page, Fig 2, the **Active variables** to be solved are specified by clicking the corresponding box. If **Comfort** is required, then the metabolic rate (W/m^2), the thermal resistance of clothing (m^2 K/W) and the water vapour pressure (Pa) must be specified; otherwise the default values will be used. Similarly, if a transient solution is required, then the total time and time step must be specified in seconds. The number of sweeps is specified in the top right hand box by either using the arrow or typing in the value then ↵. Initially, a small number of sweeps, e.g. 2, is specified and the results are checked to ensure that the problem has been correctly modelled. If so, then the problem can be run for a larger number of sweeps, otherwise the necessary changes are made and the same

procedure is repeated until the correct parameters have been set. Generally, the CPU time is directly proportional to the number of sweeps and one has to compromise between the accuracy of the solution and the execution time. There is no easy way of determining the optimum number of sweeps as this is a problem dependent quantity but generally, for room flow calculations, about 500 iterations are required for isothermal flow problems and about 800 iterations or more for non-isothermal flow solutions. However, the best way to ensure convergence is to check the residuals output. It is always possible to restart the calculations for an additional number of sweeps if convergence has not been achieved, as described in **Run Page**. For a case which has been previously run and the output files are available, a restart can be performed by clicking the **Restart** box.

The relaxation parameters for the variables to be solved are also specified in this page; however, the default values are adequate for the majority of cases solved. After all the data are input the **Apply** box is clicked to save the values.

Global Page
In this page, the case name appears in the **Case** box (up to 8 characters) and a title is given to the case in the **Title** box (up to 40 characters), see Fig 3. The room dimensions and the surface boundary conditions are specified here too. In addition to the surface temperature, the user can also specify surface heat fluxes (W/m^2). At each room surface, there is the option of specifying a convective and radiant flux components simultaneously. If a radiant flux is specified the emissivity of the surface must also be specified, otherwise a default of 0.9 is used. The radiant flux will be used in the calculation of surface-to-surface radiation exchange and the local mean radiant temperature at each computational cell. When the convective heat flux on a surface is specified as zero, the rate of heat transfer through the surface is calculated from the temperature difference between the thermal boundary layer and the surface. When the heat flux on a surface is *not* equal to zero, the temperature of the corresponding surface will not be used for the computation of the temperature in the boundary layer and beyond, but instead the heat flux, together with a default surface heat transfer coefficient, will be used for calculating the air temperature. Hence, if the heat transfer through the surface is known to be zero (i.e. an adiabatic surface), a small value for the heat flux, say, 0.001 W/m^2 instead of 0.0 W/m^2 should *be* used.

Clicking the **Apply** icon saves the data on the page.

Edit Obj Page
In this page, data for the inlets, outlets, windows, obstructions and concentration sources are specified; see Fig 4. By default there is one inlet and one outlet specified. The size and position of each object can be changed using the arrows in X,Y,Z boxes. For the inlet the velocity (perpendicular to the opening) must be specified together with the turbulence intensity, temperature and concentration (ppm) if concentration is activated. For the outlet, only the size and position is specified.

If an additional object is required then the **New** box is clicked to reveal a list of the objects as shown in Fig.4. For a window, in addition to the size and position, the temperature and heat flux in W/m^2 is specified. Similarly, for an obstacle the temperature and heat flux is specified for the whole surface of the obstacle. For an adiabatic (no heat transfer) object a small value of heat flux is specified, e.g. 0.001 W/m^2, similar to the walls. After each object is entered, the **Apply** box is clicked and when all the objects have been entered, the **Finish** box is clicked. Note up to 999 objects of each category may be specified but the more objects are specified the larger will the computation grid be.

Mesh Page
The number of grids for the calculations in the X, Y and Z direction is specified here. After the user has specified all the objects, the VORTEX pre-processor creates a default grid to fit the computation domains (Regions), see Fig 5. However, the user can alter the number of grid points in each region by clicking the arrows in each box. This process is repeated for the three co-ordinates. Finally, the

Finished box is clicked before leaving the page. If the data generated are required for a future run then the user must click the **Save** box in the **File** icon, otherwise the data will *not* be saved.

6. RUNNING VORTEX

Run Page

After the data has been provided and saved, the model can then be run by clicking the **Run** icon, which opens a new page for the main **VORTEX** program; see Fig. 6. As soon as this page is opened, **VORTEX** waits for the user to click the **Run** icon to commence computing. At any time during the execution of **VORTEX** (providing the number of sweeps has not been reached) the program can be stopped and re-run as often as required. The values of the variables at the monitoring point (**Monitor Spot Value**) are displayed and also plotted. In addition, the percentage change in the variable from the previous sweep is also printed. The monitoring spot can be changed during the execution of the program (centre of field is the default setting) by clicking the **Spot Val** in the **Graph** icon. When the new position is typed in, it can only be affected by hitting the return key ↵.

The relaxation factor, the number of iterations, the plot frequency and the number frequency of monitoring spot values and the residual errors can be changed at any time by clicking the **Solver** icon, typing the required value and hitting ↵.

Normally, the validity of the input parameters is tested by initially executing the program for a few iterations, e.g. 5, and the output checked to ensure that the problem has been modelled correctly. This is done by examining the output file **model.S** which gives a summary of the key input parameters such as surface temperatures, heat fluxes, position and size of the obstructions, inlet and outlet areas, flow rate, contaminant production rate, etc. If the initial output data is acceptable the program is then ran for a higher iteration number to obtain a converged solution. Generally, the CPU time is directly proportional to the number of iterations and one has to compromise between the accuracy of the solution and the execution time. There is no easy way of determining the optimum number of iterations as this is a problem dependent quantity but generally, for room flow calculations, about 500 iterations are required for isothermal flow problems and about 800 iterations or more for non-isothermal flow solutions.

It is always possible to restart the calculations for an additional number of iterations if convergence has not been achieved. When restarting, **VORTEX** will read all the field data produced in the previous run as well as that in **model.VDB** and use them to continue the calculations.

Output Files

When the program terminates and the output files saved on disk automatically, the results can then be visualised using **FLOWVIS** by clicking **Post Pro** in the **Tasks** icon. Details of carrying out the visualisation are described under the post-processor **FLOWVIS.**

In additional to graphic output, the user has access to numerical data via text files. Files **model.D, model.G, model.V, model.T, model.C, model.TMRT, model.PMV, model.PPD, model.TI, model.PD** and **model.AGE** represent the files for room geometry (including inlets, outlets, windows and obstructions), the computation grid in X, Y, Z co-ordinates, the velocity components U, V, W, the temperature, the concentration, the mean radiant temperature, the Predicted Mean Vote, the Predicted Percentage of Dissatisfied, the turbulence intensity, the Percentage of Dissatisfied due to draught and the age of air respectively.

The file **model.S** contains a summary of the input data and key output parameters such as the inlet velocity, inlet temperature, inlet concentration, volume flow rate, the Reynolds number and the

Archimedes number values for the inlet opening(s). It also contains mean values of the velocity, temperature, concentration, PPD and the age of air (if these are computed) for each horizontal plane in the whole field (excluding the spaces occupied by obstacles, if any). In addition, **model.S** gives mean spatial values of velocity, temperature, concentration, PPD and the age of air for the occupied zone (height of 1.8 m) and the whole space too.

File **model.R** contains the sum of the residuals for the velocity components (**U**,**V**,**W**), temperature, kinetic energy, dissipation rate and mass flow and the values of the velocity components, temperature, kinetic energy and dissipation rate at the monitored point (default at centre of the field). It also gives the distribution of the variables in the field for alternate x-y planes in the field for inspection by the user.

File **model.D** gives the room dimensions and the co-ordinates (W, E, S, N, L, R) of the supply and exit openings and obstacles. File **model.G** gives the co-ordinates of all the grids (**X(I), I** = 1 to **NI**; **Y(J)**, **J** = 1 to **NJ**; **Z(K)**, **K** = 1 to **NK**) for plotting. Note that the co-ordinates of the first and last grid points are included in the boundary values.
Files **model.ED, model.P, model.T, model.TE, model.V** and **model.VIS** store the field grid data for the turbulence dissipation, pressure, the air temperature, the turbulence energy, the three velocity components and the turbulent fluid viscosity respectively. If other variables are computed, additional output data files will also be generated. Theses files will be one or more of the following: **model.AGE, model.C, model.PD, model.PMV, model.PPD, model.TI, model.TMRT,** respectively represented the age of air, concentration of species, Predicted Dissatisfied due to draught, Predicted Mean Vote, Predicted Percentage of Dissatisfied, the turbulence intensity and the mean radiant temperature. These files contain the data arrays for graphical presentation or for further processing when restarting the computation. The arrays are dumped as:

```
for K = 1 to NK do
        for J = 1 to NJ do
                for I = 1 to NI do
                        write variable (I,J,K)
```

These output files are read and used as initial values when restarting a run to reduce program running time when a converged solution has not been reached. It should be noted that if an initial run clearly gives a diverged solution (as indicated by the residuals in Vortex Main Program) then a restart should not be performed but instead changes to the **model.VDB** data should be made before another run is carried out. These changes may include a reduction in the values of the under relaxation factors for the variable which is causing the divergence or a reduction in the size of source concentrations, such as heat fluxes.

In addition to the output files described above, **VORTEX** also generates six other files to be

used for visualisation of the results using **FLOWVIS**. These are:

model.FFX, model.FFY, model.FFZ, model.FHD, model.FST, model.GEO

The latter gives the physical co-ordinates of all the inlet and outlet openings and the obstructions and the corresponding colour code for plotting these using **FLOWVIS**. The user if desires can change the colour codes.

7. OTHER INPUT DATA FILES

File MODEL.DAT

When the **VORTEX** pre-processor is used to generate a grid, it divides each region into a uniformly spaced grid. However, the user can generate his/her own computational grid for each region in the field by generating a **model.DAT** file using an editor in the same format as required by **VORTEX**. This offers the user the flexibility for specifying his/her own grid geometry, e.g. a non-uniform grid for each or some of the regions. The format of the **model.DAT** file is given in Table 2 and the ***.DAT** files for the Examples.

After generating a specific **Model.DAT** file and saved in the directory **C:\VORTEX|WORKAREA** this can be run using the following steps:

(I) Open an existing ***.VDB** file using the Vortex Post-rocessor and click the **Run** icon.
(ii) Go to **File** icon in Vortex Main Program and select **open**.
(iii) Type the name of the file generated, e.g. **user.DAT** (without the **.DAT** extension) and and press ↵.
(iv) Click the **Open** box and the **user.DAT** file will be run normally.

File FABRIC.DAT

If the internal room surface temperatures or heat fluxes are not known **VORTEX** has the option of generating the required boundary conditions using external weather data and building fabric data. This is done assuming steady-state conduction through the fabric, i.e. heat storage is not considered. At present, the input required for defining the fabric data is via a separate input file called **FABRIC.DAT**. The structure of this data file is described below.

If the fabric heat loss/gain is to be taken into account, data are required for the wind velocity, **VAO-**, temperature, **TAO-**, emissivity of exterior wall surface, **EPSO-**, the number of structural components, **LAYER-**, thickness, **DELTA-**, and thermal conductivity, **TK-**, of each component, for each of the walls, roof and floor. The data are set in a separate file called **FABRIC.DAT** in the following format:

VAOW VAOE VAOS VAON VAOL VAOR
TAOW TAOE TAOS TAON TAOL TAOR
EPSOW EPSOE EPSOS EPSON EPSOL EPSOR

LAYERW
DELTAW(I) (I=1, LAYERW)
TKW(I) (I=1, LAYERW)

LAYERE
DELTAE(I) (I=1, LAYERE)
TKE(I) (I=1, LAYERE)

LAYERS
DELTAS(I) (I=1, LAYERS)
TKS(I) (I=1, LAYERS)

LAYERN
DELTAN(I) (I=1, LAYERN)
TKN(I) (I=1, LAYERN)

LAYERL
DELTAL(I) (I=1, LAYERL)
TKL(I) (I=1, LAYERL)

LAYERR
DELTAR(I) (I=1, LAYERR)
TKR(I) (I=1, LAYERR)

where the last characters of the parameters, **W, E, S, N, L, R,** are the west, east, south, north, left and right surfaces respectively. The thickness or thermal conductivity of each component of a wall must also be set in the same line.

In the case of a known thermal resistance, e.g. air cavity, the thickness of the layer (**DELTA-**) is taken as 1 m and the thermal conductivity (**TK-**) is taken as the reciprocal of this resistance.

The default data for the above variables are as follows:

VAO- = **VIN**
TAO- = **TIN**
EPSO- = 0.9
LAYER- = 1
DELTA- = 0.22 m
TK- = 0.84 W/mK

for the west, east, north, left and right surfaces.

For an intermediate floor or ceiling the fabric heat loss/gain is calculated in the same way as the walls. For the ground floor the equation below applies:

$$\frac{1}{U_f} = \frac{1}{U_t} + \sum \frac{x_n}{k_n} - \frac{1}{h_i} \qquad (2)$$

where

U_f = floor heat conduction coefficient excluding internal surface resistance (W/m^2K)

U_t = tabulated value of the overall heat transmission coefficient for a bare ground floor (W/m^2K)

x_n = thickness of layers above the floor, e.g. insulation carpet, etc. (m)

k_n = thermal conductivity of each layer (W/mK)

h_i = internal surface heat transfer coefficient (dependent on direction of heat flow) (W/m^2K).

The program calculates a default heat transfer coefficient for a ground floor using:

VAOS = 10000.0 (no exterior surface film resistance)
TAOS = **TIN**
EPSOS = 0.0
LAYERS = 1
TKS / DELTAS = 2.73 W/m^2K (carpet and fibrous pad)

However, a more accurate value of the heat transfer coefficient of the floor can be calculated using Equation (2) and this can be used for overwriting the default value.

If all the default values are applicable for the problem being investigated, **FABRIC.DAT** is not required. If the values for outdoor velocity and temperature are different from the default data while those for the fabric are the same as the default data, input the data for the velocity and temperature only. Thus, the input data can be simplified to:

VAOW VAOE VAOS VAON VAOL VAOR
TAOW TAOE TAOS TAON TAOL TAOR

If the emissivity of exterior wall surfaces needs to be specified as well as the velocity and temperature, the input data will become:

VAOW VAOE VAOS VAON VAOL VAOR
TAOW TAOE TAOS TAON TAOL TAOR
EPSOW EPSOE EPSOS EPSON EPSOL EPSOR

8. EXAMPLES

The following examples illustrate the use of **VORTEX**. In the first example, only the velocity and temperature distributions are required. The second example shows a more comprehensive analysis of variables, including thermal comfort analysis. The third example represents a buoyancy-driven, naturally ventilated enclosure.

Example 1

This example illustrates the use of the program for predicting the environment in an office which is cooled by supplying air from a wall at high level and extracting air from the ceiling.

The air is supply rate is 0.096 m^3/s at a speed of 2.0 m/s and temperature 12^oC to the room, which has the dimensions given below, through a diffuser situated on the middle of the west wall close to the ceiling:

LENGTH = 4.0 m HEIGHT = 2.7 m WIDTH = 3.0 m

The effective dimensions of the diffuser are 1.2 m long and 0.04 m wide. The air is extracted through a 0.4m x 0.4m ceiling opening close to the the curtain wall (east wall).

The inside temperature of each wall is assumed to be 25^oC. There is heat generation on the floor at a rate of 50 W/m^2. A window, 1.6 m wide and 1.75 m high, is situated on the east wall, 0.55 m above the floor. The temperature of the window is assumed to be 28^oC. There are two obstacles in the room, one table (at a temperature of 24^oC) and one occupant (standing position) with an assumed surface temperature of 33^oC and a metabolic heat production rate of 50 W/m^2.

The input data for this case is given in Table 3 as file **examp_1.DAT**. The of regions are automatically selected by the program, depending on the number and position of the objects in the room. A default (minimum) value for the number of grids for each region is specified but the user has the option to change the number of grid points in each region. The grid chosen for this example is 31* 26 *32. The finer the grid the more accurate the solution will be but this is at the expense of a longer run time. Generally, if the grid size in each direction is doubled it will take 8 times longer to run the problem for the same number of iterations.

A theoretical investigation has been carried out to optimise the distance of the nearest grid point from a solid surface and was found that this distance should be set to about 0.01 m to achieve a region which is most appropriate for the application of the turbulent flow wall functions used in **VORTEX**.

The computation results for this case are presented graphically in Figures 7 and 8 as velocity vectors and temperature contours for two planes. One vertical plane through the supply diffuser and the other one is a horizontal plane. The values of the velocity components, temperature, etc. for each grid point can be found in file **examp_1.R** and a summary result of the output in File **examp_1.S** (see Table 4).

Example 2

This is a heating example for the same office room described in Example 1 with a square diffuser at the centre of the ceiling discharging the air radially. The supply air flow rate is 0.09 m^3/s. The size of the square diffuser is 0.6 m x 0.6 m x 0.1 m effective slot width. The supply air temperature is 30^oC. This room is on the ground floor of a building which has three external walls, one internal wall (west wall) and a ceiling adjoining the floor of the next level. The internal temperatures of the ceiling and the west wall are known to be both 21^oC, whereas the internal temperatures of the floor and the other three walls are not known. However, the outdoor design temperature is given as 1^oC and the design wind speed is 3.4 m/s. The single-glazed window has an internal temperature of 10^oC which is calculated from the U-value, the outdoor temperature and the estimated room temperature of 24^oC. There are also heat losses from the curtain wall of 20 W/m^2 and from the floor of 25 W/m^2.

The room is constructed using standard building materials, i.e. the default values of fabric properties given in file **FABRIC.DAT** are assumed to apply here. In addition to calculating the heat loss through the external room surfaces to determine their internal surface temperature, this example also analyses the thermal comfort in the room.

The file **examp_2.DAT** for this example is given in Table 5. Only minor changes have been made to the file for Example 1 in Table 3. The **Comfort** and **Conduction** boxes are both selected for the computation of thermal comfort and conduction heat transfer through the external surfaces. The opening area (0.3 x 0.3) = 0.09 m^2. For the give flow rate of 0.09 m^3/s the supply air velocity is 1 m/s at the inlet (duct) position. However, this is not the supply velocity to the room as the diffuser will diffuse the air in a radial direction over the ceiling.

To represent this effect a thin baffle plate (obstacle) 0.6 m x 0.6 m is positioned 0.1m below the inlet opening to provide a radial diffusion of the supply air and effectively representing the action of the diffuser. This is of course a simplification to the complex aerodynamics of a ceiling diffuser but if the gap between the baffle and the ceiling is set to produce the same supply velocity as the actual diffuser this approximation should produce reliable results.

The setting of room surface temperature is no longer critical for this example because the true values are calculated using the heat conduction equations through the fabric. Therefore, only approximate values for the internal surface temperatures need to be specified.

File **FABRIC.DAT** is created to input the outdoor environmental conditions for the calculation of fabric heat transfer with the following data:

```
3.4   3.4   10000.0    0.0    0.0    3.4
1.0   1.0      10.0   24.0   24.0    1.0
```

The results of the computation are shown plotted in Figures 9 to 11 for velocity, air temperature, mean radiant temperature and predicted percentage of dissatisfied. Table 6 shows the summary output data for the problem in the form of file **examp_2.S**

Example 3

This example deals with a large enclosure (e.g. atrium) which is naturally ventilated by an open door and two top outlets on either side of the structure. The air enters through the door and rises up by the buoyancy resulting from the heat flux on the roof and the floor (e.g. solar gain). It is assumed that the other four surfaces are adiabatic. There are two platforms just below the outlets running across the whole building and 2 m wide and 0.15m thick.

The internal dimensions of the enclosure are:

LENGTH = 12.0 m HEIGHT = 10.0 m WIDTH = 8.0 m

The outdoor air temperature is assumed to be 22.0^oC and the heat fluxes on the roof and the floor are 50 and 25 W/m^2 respectively. The door is 2.0 m high and 1.5 m wide and is located on the west wall. The two outlets are 0.8 m high and 1.5 m wide each and are located at the top of the west and east walls. It is also assumed that there are 50 people occupying the building each producing 4.7 ml/s of CO_2.

To activate the calculation for buoyancy the inlet velocity must be specified as 0.01 m/s but the true inlet velocity will be calculated by the program and updated during each iteration. The heat fluxes on the roof and floor are specified and an estimate is made of the temperature for these two surfaces. However, these temperatures will not be used in the calculations as the heat flux has a precedent over surface temperatures. To represent an adiabatic wall, a very small heat flux (e.g. 0.001) is specified for the wall and an approximate wall surface temperature is specified. This is the case for the four adiabatic walls in this example.

The 50 CO_2 concentration sources are spread over two regions to include 50 node points and the emission for each node is specified in ml/s. These are point sources which are determined by the number of grid points selected for this region in each direction. For each of the two regions, only one grid point is selected for the y direction, 5 grid points are selected in each of the x and z direction giving a total number of source point for the region of 1*5*5 = 25. The total CO_2 emission is 0.235 l/s. **examp_3 .DAT** for this example is given in Table 7.

The results of the computation for velocity, temperature and CO_2 concentration in ppm are plotted in Figures 12 and 13 for two planes across the building. Table 8 shows a summary output.

Table 1. Source terms of the flow equations

Equation	ϕ	Γ_ϕ	S_ϕ
Continuity	1	0	0
U-momentum	U	μ_e	$-\frac{\partial p}{\partial x} + \frac{\partial}{\partial x}(\mu_e \frac{\partial U}{\partial x}) + \frac{\partial}{\partial y}(\mu_e \frac{\partial V}{\partial x}) + \frac{\partial}{\partial z}(\mu_e \frac{\partial W}{\partial x}) - \frac{2}{3}\frac{\partial}{\partial x}(\rho k)$
V-momentum	V	μ_e	$-\frac{\partial p}{\partial y} + \frac{\partial}{\partial x}(\mu_e \frac{\partial U}{\partial y}) + \frac{\partial}{\partial y}(\mu_e \frac{\partial V}{\partial y}) + \frac{\partial}{\partial z}(\mu_e \frac{\partial W}{\partial y}) - \frac{2}{3}\frac{\partial}{\partial y}(\rho k) - g(\rho - \rho_o)$
W-momentum	W	μ_e	$-\frac{\partial p}{\partial z} + \frac{\partial}{\partial x}(\mu_e \frac{\partial U}{\partial z}) + \frac{\partial}{\partial y}(\mu_e \frac{\partial V}{\partial z}) + \frac{\partial}{\partial z}(\mu_e \frac{\partial W}{\partial z}) - \frac{2}{3}\frac{\partial}{\partial y}(\rho k)$
Temperature	T	Γ_e	q / C_p
Concentration	C	Γ_c	S_c
Kinetic energy	k	Γ_k	$G_s - C_D \rho \varepsilon + G_B$
Dissipation rate	ε	Γ_ε	$C_1 \frac{\varepsilon}{k}(G_s + G_B) - C_2 \rho \frac{\varepsilon^2}{k}$

Notes: $\mu_e = \mu + \mu_t$ $\quad \mu_t = C_\mu \rho k^2/\varepsilon$ $\quad \sigma_\varepsilon = \kappa^{2/}/(C_2 - C_1)C_\mu^{1/2}$

$\Gamma_e = \mu_t/\sigma_t + \mu/\sigma_l$ $\quad \Gamma_c = \mu_t/\sigma_c + \mu/\sigma_l$

$\Gamma_k = \mu_e/\sigma_k$ $\quad \Gamma_\varepsilon = \mu_e/\sigma_\varepsilon$

Kinetic energy generation by shear

$$G_s = \mu_t \{2[(\frac{\partial U}{\partial x})^2 + (\frac{\partial V}{\partial y})^2 + (\frac{\partial W}{\partial z})^2] + (\frac{\partial U}{\partial y} + \frac{\partial V}{\partial x})^2 + (\frac{\partial U}{\partial z} + \frac{\partial W}{\partial x})^2 + (\frac{\partial V}{\partial z} + \frac{\partial W}{\partial y})^2\}$$

Kinetic energy generation by buoyancy

$$G_B = \beta g \frac{\mu_t}{\sigma_t}\frac{\partial T}{\partial y}$$

Empirical constants in the equations	C_μ	C_D	C_1	C_2	σ_c	σ_k	σ_t
	0.09	1.0	1.44	1.92	0.9	1.0	0.9

Table 2. Structure of MODEL.DAT (comments in lower case)

Variable List	Summary of Variable List
ICONC, ICOMF, IDRA, IRADI, ICOND,IAGE	Control flags for concentration, comfort, draught, Radiation, conduction and age
L, H, W	Length, Height & Width
NI, NJ, NK	Maximum grid number in x (length), y (Height),z(Width) Directions
I, X(I)	Grid and corresponding coordinate in x direction (I=1 to NI)
J, Y(J)	Grid and corresponding coordinate in y direction (J=1 to NJ)
K, Z(K)	Grid and corresponding coordinate in z direction(K=1to NK)
NIN	Number of air supply openings
IINW(I), IINE(I), JINS(I), JINN(I), KINL(I), KINR(I)	Coordinates of supply opening I (I=1 to NIN)
NOUT	Number of air exits
IOUTW(I), IOUTE(I), JOUTS(I), JOUTN(I), KOUTL(I), KOUTR(I)	Coordinates of exit opening I (I=1 to NOUT)
NWIN	Number of windows
IWINW(I), IWINE(I), JWINS(I), JWINN(I),KWINL(I), KWINR(I), TWIN(I), HWIN(I)	Coordinates, temperature and heat flux (W/m2)of window I (I=1 to NWIN)
NOBS	Number of obstacles
IOBSW(I), IOBSE(I), JOBSS(I), JOBSN(I), KOBSL(I), KOBSR(I), TOBS(I), HOBS(I)	Coordinates, temperature and heat production (W/m^2) of obstacle I (I=1 to NOBS)
NCON	Number of concentration sources
ICONW(I), ICONE(I), JCONS(I), JCONN(I), KCONL(I), KCONR(I), C(I)	Coordinates and generation rate (ml/s or mg/s) of concentration source I (I=1 to NCONC)
TW, TE, TS, TN, TL, TR	Wall surface temperatures
HCW, HCE, HCS, HCN, HCL, HCR	Convective heat fluxes on room surfaces
HRW, HRE, HRS, HRN, HRL, HRR	Radiant heat fluxes on surfaces
EPSW, EPSE, EPSS, EPSN, EPSL, EPSR	Emissivity of interior wall surfaces
UIN, VIN, WIN, DC, TURBIN, TIN, CIN	U, V and W Velocity, Discharge coefficient, turbulence intensity, temperature and concentration of supply air
MAXIT	Maximum number of iterations
URFU, URFV, URFW, URFP, URFK, URFE, URFVIS, URFT, URFC, URFG	Under relaxation factors
M, ICLO, PA	Metabolic rate, thermal resistance of clothing and partial water vapour pressure
NP	Number of points for output data
X(I), Y(I), Z(I)	Coordinates of point I (I=1 to NP)

Table 3. File examp_1.DAT for Example 1

```
Wall Supply
EXAMP_1

0  0  0  0  0
4.00000e+000   2.70000e+000   3.00000e+000
31  25  32

 1  -1.000000e-002
 2   1.000000e-002
 3   1.466667e-001
 4   4.000000e-001
 5   6.533333e-001
 6   7.900000e-001
 7   8.100000e-001
 8   9.800000e-001
 9   1.300000e+000
10  1.620000e+000
11  1.790000e+000
12  1.810000e+000
13  1.860000e+000
14  1.940000e+000
15  1.990000e+000
16  2.010000e+000
17  2.135000e+000
18  2.365000e+000
19  2.490000e+000
20  2.510000e+000
21  2.680000e+000
22  3.000000e+000
23  3.319999e+000
24  3.489999e+000
25  3.509999e+000
26  3.699999e+000
27  3.889999e+000
28  3.909999e+000
29  3.949999e+000
30  3.990000e+000
31  4.010000e+000

 1  -1.000000e-002
 2   1.000000e-002
 3   1.475000e-001
 4   4.024999e-001
 5   5.399998e-001
 6   5.599998e-001
 7   5.999999e-001
 8   6.400000e-001
 9   6.600000e-001
10  7.000000e-001
11  7.400000e-001
12  7.600000e-001
13  1.010000e+000
14  1.490000e+000
15  1.740000e+000
16  1.760000e+000
17  1.897500e+000
18  2.152500e+000
19  2.290000e+000
20  2.310000e+000
```

21 2.480000e+000
22 2.650000e+000
23 2.670000e+000
24 2.690000e+000
25 2.710000e+000

 1 -1.000000e-002
 2   1.000000e-002
 3   1.850000e-001
 4   5.150000e-001
 5   6.900001e-001
 6   7.100000e-001
 7   8.000001e-001
 8   8.900001e-001
 9   9.100001e-001
10 1.018750e+000
11 1.127500e+000
12 1.147500e+000
13 1.218750e+000
14 1.290000e+000
15 1.310000e+000
16 1.468750e+000
17 1.627500e+000
18 1.647500e+000
19 1.668750e+000
20 1.690000e+000
21 1.710000e+000
22 1.810000e+000
23 1.990000e+000
24 2.090000e+000
25 2.110000e+000
26 2.200000e+000
27 2.290001e+000
28 2.310001e+000
29 2.485001e+000
30 2.815000e+000
31 2.990000e+000
32 3.010000e+000

1
1 1 23 24 9 24

1
25 27 25 25 15 20

1
31 31 6 19 6 27   2.80000e+001   0.00000e+000

2
 7 11 9 11   9 24   2.40000e+001 0.00000e+000
16 19 2 15 12 17   3.30000e+001 5.00000e+001

0

2.50000e+001 2.50000e+001 2.50000e+001 2.50000e+001 2.50000e+001 2.50000e+001
0.00000e+000 0.00000e+000 5.00000e+001 0.00000e+000 0.00000e+000 0.00000e+000

2.00000e+000   4.99997e-002   1.20000e+001

900

5.00000e-001
5.00000e-001
5.00000e-001
8.00000e-001
5.00000e-001
5.00000e-001
8.00000e-001
5.00000e-001

0

0 : TRANSIENT -> TOTAL_TIME, STEP_SIZE

0 : RESTART -> CONC, COMFORT, DRAUGHT, RAD, CONDUCT, AGE

Table 4 File examp_1.S for Example 1

Wall Supply
EXAMP_1

* ROOM AIR MOVEMENT *

ROOM DIMENSIONS (m)

LENGTH = 4.0000
WIDTH = 3.0000
HEIGHT = 2.7000

FLUID INLET (m)

INLET 1
WIDTH = 1.2000 , HEIGHT = 0.0400 , AREA = 0.0480
TOTAL INLET AREA = 0.0480 m**2

FLUID OUTLET (m)

OUTLET 1
WIDTH = 0.4000 , HEIGHT = 0.4000 , AREA = 0.1600
TOTAL OUTLET AREA = 0.1600 m**2

OBSTACLE SIZE (m)

OBSTACLE 1
X-DIRECTION: X1 = 0.8000 , X2 = 1.8000 , LENGTH = 1.0000
Y-DIRECTION: Y1 = 0.6500 , Y2 = 0.7500 , HEIGHT = 0.1000
Z-DIRECTION: Z1 = 0.9000 , Z2 = 2.1000 , WIDTH = 1.2000

OBSTACLE 2
X-DIRECTION: X1 = 2.0000 , X2 = 2.5000 , LENGTH = 0.5000
Y-DIRECTION: Y1 = 0.0000 , Y2 = 1.7500 , HEIGHT = 1.7500
Z-DIRECTION: Z1 = 1.1375 , Z2 = 1.6375 , WIDTH = 0.5000

WINDOWS (m)

WINDOW 1
WIDTH = 1.6000 , HEIGHT = 1.7500 , AREA = 2.8000
TOTAL AREA OF WINDOWS = 2.8000 m**2

INLET FLUID PROPERTIES

VELOCITY	=	2.000E+0 m/s
TURBULENCE INTENSITY	=	5.000E-2
TEMPERATURE	=	1.200E+1 deg.C
DENSITY	=	1.232E+0 kg/m**3
LAMINAR VISCOSITY	=	1.771E-5 Pa s
AIR FLOW RATE	=	9.600E-2 m**3/s
REYNOLDS NUMBER	=	5.567E+3
PRANDTL NUMBER	=	7.050E-1

INITIAL INSIDE SURFACE TEMPERATURE (deg. C)

WEST WALL	=	25.0000
EAST WALL	=	25.0000
FLOOR	=	25.0000
CEILING	=	25.0000
LEFT WALL	=	25.0000
RIGHT WALL	=	25.0000
WINDOW 1	=	28.0000
OBSTACLE 1	=	24.0000
OBSTACLE 2	=	33.0000

CONVECTIVE HEAT FLUX ON SURFACES (W/m**2)

WEST WALL	=	0.0000
EAST WALL	=	0.0000
FLOOR	=	50.0000
CEILING	=	0.0000
LEFT WALL	=	0.0000
RIGHT WALL	=	0.0000
WINDOW 1	=	0.0000
OBSTACLE 1	=	0.0000
OBSTACLE 2	=	50.0000

ARCHIMEDES NUMBER = 0.337E-2

MEAN VALUES AT DIFFERENT HEIGHT LEVELS

GRID	HEIGHT (m)	VELOCITY (m/s)	TEMPERATURE (deg.C)
2	0.0100	0.2643	25.2267
3	0.1475	0.2261	23.7670
4	0.4025	0.1891	23.7162
5	0.5400	0.1743	23.7878
6	0.5600	0.1713	23.8006
7	0.6000	0.1666	23.8202
8	0.6400	0.1633	23.8364
9	0.6600	0.1690	23.7377
10	0.7000	0.1633	23.7459
11	0.7400	0.1582	23.7498
12	0.7600	0.1455	23.7483
13	1.0100	0.1241	23.7846
14	1.4900	0.1296	23.8584
15	1.7400	0.1368	23.8906
16	1.7600	0.1352	23.9483
17	1.8975	0.1495	23.9587
18	2.1525	0.1765	23.8478
19	2.2900	0.1929	23.6041
20	2.3100	0.1981	23.5301

21	2.4800	0.2197	23.3750
22	2.6500	0.3233	22.8203
23	2.6700	0.4580	22.3344
24	2.6900	0.4647	22.4381

MEAN VALUES IN THE OCCUPIED ZONE
(from floor to height 1.8 m)

AIR VELOCITY = 0.2643 m/s
AIR TEMPERATURE = 25.2267 deg.C

MEAN VALUES IN THE SPACE

AIR VELOCITY = 0.1762 m/s
AIR TEMPERATURE = 23.7598

Table 5 File examp_2.DAT for Example 2

```
Ceiling supply
EXAMP_2

0  1  0  0  1
4.00000e+000   2.70000e+000   3.00000e+000
38  34  37

 1  -1.000000e-002
 2   1.000000e-002
 3   1.466667e-001
 4   4.000000e-001
 5   6.533334e-001
 6   7.900001e-001
 7   8.100001e-001
 8   9.633334e-001
 9   1.250000e+000
10  1.536667e+000
11  1.690000e+000
12  1.710000e+000
13  1.750000e+000
14  1.790000e+000
15  1.810000e+000
16  1.825000e+000
17  1.840000e+000
18  1.860000e+000
19  1.925000e+000
20  1.990000e+000
21  2.010000e+000
22  2.075000e+000
23  2.140000e+000
24  2.160000e+000
25  2.225000e+000
26  2.290000e+000
27  2.310000e+000
28  2.346667e+000
29  2.400000e+000
30  2.453334e+000
31  2.490000e+000
32   2.510000e+000
33  2.702500e+000
34  3.067500e+000
35  3.432500e+000
36  3.797500e+000
37  3.990000e+000
38  4.010000e+000

 1  -1.000000e-002
 2   1.000000e-002
 3   1.099999e-001
 4   2.899998e-001
 5   3.899998e-001
 6   4.099998e-001
 7   4.749999e-001
 8   5.400000e-001
 9   5.600000e-001
10  6.000000e-001
11  6.400000e-001
12  6.600000e-001
13  7.000000e-001
```

14 7.400000e-001
15 7.600000e-001
16 9.300000e-001
17 1.250000e+000
18 1.570000e+000
19 1.740000e+000
20 1.760000e+000
21 1.897500e+000
22 2.152500e+000
23 2.290000e+000
24 2.310000e+000
25 2.400000e+000
26 2.490000e+000
27 2.510000e+000
28 2.550000e+000
29 2.590000e+000
30 2.610000e+000
31 2.635000e+000
32 2.665000e+000
33 2.690000e+000
34 2.710000e+000

1 -1.000000e-002
2 1.000000e-002
3 1.850000e-001
4 5.150000e-001
5 6.900001e-001
6 7.100000e-001
7 8.000001e-001
8 8.900001e-001
9 9.100001e-001
10 1.050000e+000
11 1.190000e+000
12 1.210000e+000
13 1.225000e+000
14 1.240000e+000
15 1.260000e+000
16 1.300000e+000
17 1.340000e+000
18 1.360000e+000
19 1.500000e+000
20 1.640000e+000
21 1.660000e+000
22 1.700000e+000
23 1.740000e+000
24 1.760000e+000
25 1.775000e+000
26 1.790000e+000
27 1.810000e+000
28 1.950000e+000
29 2.090000e+000
30 2.110000e+000
31 2.200001e+000
32 2.290001e+000
33 2.310001e+000
34 2.485001e+000
35 2.815000e+000
36 2.990000e+000
37 3.010000e+000

```
18  23  34  34  18  20

1
1  1  2  5  18  23

1
38  38  9  23  6  32   1.00000e+001   0.00000e+000

3
12  26  27  29  12  26   2.80000e+001   0.00000e+000
 7  14  12  14    9  29   2.40000e+001   1.00000e-002
21  31   2  19  15   23   3.30000e+001   5.00000e+001

0

2.10000e+001       2.10000e+001         2.10000e+001  2.10000e+001   2.10000e+001          2.10000e+001
0.00000e+000      -2.00000e+001        -2.50000e+001  0.00000e+000   0.00000e+000   0.00000e+000

1.00000e+000   4.99997e-002   2.80000e+001

800
5.00000e-001
5.00000e-001
5.00000e-001
8.00000e-001
5.00000e-001
5.00000e-001
8.00000e-001
5.00000e-001

7.00000e+001   1.10000e-001   1.50000e+003

0

0           :  TRANSIENT -> TOTAL_TIME, STEP_SIZE

0           : RESTART -> CONC, COMFORT, DRAUGHT, RAD,CONDUCT, AGE
```

Table 6 File examp_2.S for Example 2

Ceiling supply
EXAMP_2

* ROOM AIR MOVEMENT *

ROOM DIMENSIONS (m)

LENGTH = 4.0000
WIDTH = 3.0000
HEIGHT = 2.7000

FLUID INLET (m)

INLET 1
WIDTH = 0.3000 , HEIGHT = 0.3000 , AREA = 0.0900
TOTAL INLET AREA = 0.0900 m**2

FLUID OUTLET (m)

OUTLET 1
WIDTH = 0.4000 , HEIGHT = 0.4000 , AREA = 0.1600
TOTAL OUTLET AREA = 0.1600 m**2

OBSTACLE SIZE (m)

OBSTACLE 1
X-DIRECTION: X1 = 1.7000 , X2 = 2.3000 , LENGTH = 0.6000
Y-DIRECTION: Y1 = 2.5000 , Y2 = 2.6000 , HEIGHT = 0.1000
Z-DIRECTION: Z1 = 1.2000 , Z2 = 1.8000 , WIDTH = 0.6000

OBSTACLE 2
X-DIRECTION: X1 = 0.8000 , X2 = 1.8000 , LENGTH = 1.0000
Y-DIRECTION: Y1 = 0.6500 , Y2 = 0.7500 , HEIGHT = 0.1000
Z-DIRECTION: Z1 = 0.9000 , Z2 = 2.1000 , WIDTH = 1.2000

OBSTACLE 3
X-DIRECTION: X1 = 2.0000 , X2 = 2.5000 , LENGTH = 0.5000
Y-DIRECTION: Y1 = 0.0000 , Y2 = 1.7500 , HEIGHT = 1.7500
Z-DIRECTION: Z1 = 1.2500 , Z2 = 1.7500 , WIDTH = 0.5000

WINDOWS (m)

WINDOW 1
WIDTH = 1.6000 , HEIGHT = 1.7500 , AREA = 2.8000
TOTAL AREA OF WINDOWS = 2.8000 m**2

INLET FLUID PROPERTIES

VELOCITY	=	1.000E+0 m/s
TURBULENCE INTENSITY	=	5.000E-2
TEMPERATURE	=	2.800E+1 deg.C
DENSITY	=	1.167E+0 kg/m**3
LAMINAR VISCOSITY	=	1.848E-5 Pa s
AIR FLOW RATE	=	9.000E-2 m**3/s
REYNOLDS NUMBER	=	1.894E+4
PRANDTL NUMBER	=	7.050E-1

INITIAL INSIDE SURFACE TEMPERATURE (deg. C)

WEST WALL	=	21.0000
EAST WALL	=	21.0000
FLOOR	=	21.0000
CEILING	=	21.0000
LEFT WALL	=	21.0000

RIGHT WALL = 21.0000
WINDOW 1 = 10.0000
OBSTACLE 1 = 28.0000
OBSTACLE 2 = 24.0000
OBSTACLE 3 = 33.0000

CONVECTIVE HEAT FLUX ON SURFACES (W/m**2)

WEST WALL = 0.0000
EAST WALL = -20.0000
FLOOR = -25.0000
CEILING = 0.0000
LEFT WALL = 0.0000
RIGHT WALL = 0.0000
WINDOW 1 = 0.0000
OBSTACLE 1 = 0.0000
OBSTACLE 2 = 0.0100
OBSTACLE 3 = 50.0000

ARCHIMEDES NUMBER = 0.588E-1

MEAN VALUES AT DIFFERENT HEIGHT LEVELS

GRID	HEIGHT (m)	VELOCITY (m/s)	TEMPERATURE (deg.C)	PPD (%)
2	0.0100	0.0411	18.8165	44.2161
3	0.1100	0.0390	20.1507	28.6887
4	0.2900	0.0361	20.7998	23.9167
5	0.3900	0.0334	21.0946	21.4450
6	0.4100	0.0320	21.1576	20.7442
7	0.4750	0.0294	21.3260	19.6995
8	0.5400	0.0275	21.4893	18.5629
9	0.5600	0.0269	21.5429	18.2330
10	0.6000	0.0260	21.6395	17.7423
11	0.6400	0.0250	21.7364	17.1284
12	0.6600	0.0237	21.8538	16.6565
13	0.7000	0.0227	21.9597	16.1703
14	0.7400	0.0222	22.0629	15.5793
15	0.7600	0.0216	22.1702	15.1046
16	0.9300	0.0177	22.5210	13.5197
17	1.2500	0.0159	23.1077	11.0114
18	1.5700	0.0171	23.6641	8.9385
19	1.7400	0.0165	23.9666	7.8009
20	1.7600	0.0165	24.0508	7.4471
21	1.8975	0.0169	24.2727	6.9226
22	2.1525	0.0230	24.7394	6.0424
23	2.2900	0.0299	25.0394	5.6205
24	2.3100	0.0318	25.0988	5.3816
25	2.4000	0.0335	25.2977	5.3065
26	2.4900	0.0335	25.5273	5.2744
27	2.5100	0.0362	25.6149	5.2921
28	2.5500	0.0500	25.7635	5.3485
29	2.5900	0.0834	25.9620	5.3707
30	2.6100	0.1514	26.1898	5.3750
31	2.6350	0.1647	26.3393	5.3890
32	2.6650	0.1691	26.4723	5.4389
33	2.6900	0.1628	26.5418	7.9241

MEAN VALUES IN THE OCCUPIED ZONE

(from floor to height 1.8 m)

AIR VELOCITY	=	0.0411 m/s
AIR TEMPERATURE	=	18.8165 deg.C
PREDICTED MEAN VOTE	=	-1.4005
PREDICTED PERCENTAGE OF DISSATISFIED	=	44.2161 %

MEAN VALUES IN THE SPACE

AIR VELOCITY	=	0.0303 m/s
AIR TEMPERATURE	=	23.1789 deg.C
MEAN RADIANT TEMPERATURE	=	18.9988 deg.C
PREDICTED MEAN VOTE	=	-0.4815
PREDICTED PERCENTAGE OF DISSATISFIED	=	12.7640 %

TEMPERATURE AT THE EXIT	=	0.2198E+2 deg.C
VENTILATION EFFICIENCY	=	0.6552E+2 %
PREDICTED PERCENTAGE OF DISSATISFIED	=	0.4422E+2 %

Table 7 File examp_3.DAT for Example 3

```
Naturally Ventilated Atrium
Examp_3

1  0  0  0  0
1.20000e+001   1.00000e+001   8.00000e+000
27  29  20

 1  -1.000000e-002
 2   1.000000e-002
 3   3.466667e-001
 4   1.000000e+000
 5   1.653334e+000
 6   1.990000e+000
 7   2.010000e+000
 8   2.430000e+000
 9   3.250000e+000
10  4.070000e+000
11  4.490000e+000
12  4.510000e+000
13  4.912500e+000
14  5.697500e+000
15  6.482500e+000
16  7.267500e+000
17  8.052500e+000
18  8.837500e+000
19  9.240000e+000
20  9.260000e+000
21  9.625000e+000
22  9.990000e+000
23  1.001000e+001
24  1.051000e+001
25  1.149000e+001
26  1.199000e+001
27  1.201000e+001

 1  -1.000000e-002
 2   1.000000e-002
 3   2.025000e-001
 4   5.675000e-001
 5   9.325000e-001
 6   1.297500e+000
 7   1.490000e+000
 8   1.525000e+000
 9   1.560000e+000
10  1.672500e+000
11  1.877500e+000
12  1.990000e+000
13  2.010000e+000
14  2.520714e+000
15  3.522143e+000
16  4.523571e+000
17  5.525000e+000
18  6.526429e+000
19  7.527857e+000
20  8.529285e+000
21  9.040000e+000
22  9.060000e+000
23  9.125000e+000
24  9.189999e+000
25  9.209999e+000
26  9.409999e+000
27  9.789999e+000
```

```
28  9.989999e+000
29  1.001000e+001

 1  -1.000000e-002
 2   1.000000e-002
 3   3.750000e-001
 4   7.400000e-001
 5   7.600000e-001
 6   1.385000e+000
 7   2.615000e+000
 8   3.240000e+000
 9   3.260000e+000
10  3.635000e+000
11  4.365000e+000
12  4.740000e+000
13  4.760000e+000
14  5.385000e+000
15  6.615000e+000
16  7.240000e+000
17  7.260000e+000
18  7.625000e+000
19  7.990000e+000
20  8.010000e+000

1
1  1  2  12  9  12

2
1  1  25  28  9  12
27  27  25  28  9  12

0

2
  2    6  22  24  2  19   2.50000e+001   1.00000e-003
23  26  22  24  2  19   2.50000e+001   1.00000e-003

2
12  19  8  8    5    8   4.70000e+000
12  19  8  8  13  16   4.70000e+000

2.50000e+001   2.50000e+001  2.50000e+001   2.50000e+001   2.50000e+001   2.50000e+001
1.00000e-003   1.00000e-003    2.50000e+001   5.00000e+001   1.00000e-003   1.00000e-003

1.00000e-002   5.00000e-002   2.20000e+001   3.50000e+002

1200
5.00000e-001
5.00000e-001
5.00000e-001
8.00000e-001
5.00000e-001
5.00000e-001
8.00000e-001
5.00000e-001
8.00000e-001

0

0            : TRANSIENT -> TOTAL_TIME, STEP_SIZE

0           : RESTART -> CONC, COMFORT, DRAUGHT, RAD, CONDUCT,                AGE
```

Table 8 File examp_3.S for Example 3

Naturally Ventilated Atrium
EXAMP_3

* ROOM AIR MOVEMENT *

ROOM DIMENSIONS (m)

LENGTH = 12.0000
WIDTH = 8.0000
HEIGHT = 10.0000

FLUID INLET (m)

INLET 1
WIDTH = 1.5000 , HEIGHT = 2.0000 , AREA = 3.0000
TOTAL INLET AREA = 3.0000 m**2

FLUID OUTLET (m)

OUTLET 1
WIDTH = 1.5000 , HEIGHT = 0.8000 , AREA = 1.2000

OUTLET 2
WIDTH = 1.5000 , HEIGHT = 0.8000 , AREA = 1.2000
TOTAL OUTLET AREA = 2.4000 m**2

OBSTACLE SIZE (m)

OBSTACLE 1
X-DIRECTION: X1 = 0.0000 , X2 = 2.0000 , LENGTH = 2.0000
Y-DIRECTION: Y1 = 9.0500 , Y2 = 9.2000 , HEIGHT = 0.1500
Z-DIRECTION: Z1 = 0.0000 , Z2 = 8.0000 , WIDTH = 8.0000

OBSTACLE 2
X-DIRECTION: X1 = 10.0000 , X2 = 12.0000 , LENGTH = 2.0000
Y-DIRECTION: Y1 = 9.0500 , Y2 = 9.2000 , HEIGHT = 0.1500
Z-DIRECTION: Z1 = 0.0000 , Z2 = 8.0000 , WIDTH = 8.0000

CONCENTRATION SOURCE SIZE (m)

CONCENTRATION SOURCE 1
X-DIRECTION: X1 = 4.5000 , X2 = 9.2500 , LENGTH = 4.7500
Y-DIRECTION: Y1 = 1.5250 , Y2 = 1.5250 , HEIGHT = 0.0000
Z-DIRECTION: Z1 = 0.7500 , Z2 = 3.2500 , WIDTH = 2.5000
VOLUME = 0.00 m**3
CONCENTRATION = 4.70E-6 m**3/s

CONCENTRATION SOURCE 2
X-DIRECTION: X1 = 4.5000 , X2 = 9.2500 , LENGTH = 4.7500
Y-DIRECTION: Y1 = 1.5250 , Y2 = 1.5250 , HEIGHT = 0.0000
Z-DIRECTION: Z1 = 4.7500 , Z2 = 7.2500 , WIDTH = 2.5000
VOLUME = 0.00 m**3
CONCENTRATION = 4.70E-6 m**3/s

INITIAL INSIDE SURFACE TEMPERATURE (deg. C)

WEST WALL	=	25.0000
EAST WALL	=	25.0000
FLOOR	=	25.0000
CEILING	=	25.0000
LEFT WALL	=	25.0000
RIGHT WALL	=	25.0000
OBSTACLE 1	=	25.0000
OBSTACLE 2	=	25.0000

CONVECTIVE HEAT FLUX ON SURFACES (W/m**2)

WEST WALL	=	0.0010
EAST WALL	=	0.0010
FLOOR	=	25.0000
CEILING	=	50.0000
LEFT WALL	=	0.0010
RIGHT WALL	=	0.0010
OBSTACLE 1	=	0.0010
OBSTACLE 2	=	0.0010

INLET FLUID PROPERTIES

VELOCITY	=	8.981E-1 m/s
TURBULENCE INTENSITY	=	5.000E-2
TEMPERATURE	=	2.200E+1 deg.C
DENSITY	=	1.191E+0 kg/m**3
LAMINAR VISCOSITY	=	1.820E-5 Pa s
AIR FLOW RATE	=	2.694E+0 m**3/s
REYNOLDS NUMBER	=	1.007E+5
PRANDTL NUMBER	=	7.050E-1
CONCENTRATION	=	350.0000 *E-6

TOTAL CONCENTRATION SOURCES = 300.800E-6 m**3/s

ARCHIMEDES NUMBER = 0.259E+0

MEAN VALUES AT DIFFERENT HEIGHT LEVELS

GRID	HEIGHT (m)	VELOCITY (m/s)	TEMPERATURE (deg.C)	CONC (*E-6)
2	0.0100	0.3015	23.4311	386.3337
3	0.2025	0.3264	22.6553	391.0672
4	0.5675	0.2934	22.4574	404.3911
5	0.9325	0.2563	22.4352	421.9599
6	1.2975	0.2232	22.4589	445.5610
7	1.4900	0.2100	22.4771	464.7589
8	1.5250	0.2036	22.4806	469.2646
9	1.5600	0.1964	22.4832	466.5956
10	1.6725	0.2031	22.4923	459.9988
11	1.8775	0.1874	22.5117	453.9204
12	1.9900	0.1499	22.5236	453.0828
13	2.0100	0.1433	22.5245	453.4329
14	2.5207	0.1167	22.5622	455.3239
15	3.5221	0.1146	22.5878	455.4532
16	4.5236	0.1128	22.6131	452.1322
17	5.5250	0.1123	22.6392	449.1729
18	6.5264	0.1114	22.6680	447.7545
19	7.5279	0.1116	22.7088	447.2695
20	8.5293	0.1182	22.8274	445.7350
21	9.0400	0.1246	23.0479	441.5130
22	9.0600	0.0918	23.1795	446.7566
23	9.1250	0.0908	23.2324	446.7189

24	9.1900	0.0918	23.2891	446.7066
25	9.2100	0.1419	23.3458	437.9984
26	9.4100	0.1562	23.5355	443.1207
27	9.7900	0.1725	24.2977	444.9369
28	9.9900	0.1712	26.3969	446.3040

MEAN VALUES IN THE OCCUPIED ZONE
(from floor to height 1.8 m)

AIR VELOCITY	= 0.2602 m/s
AIR TEMPERATURE	= 22.5479 deg.C
CONCENTRATION	= 424.4172 *E-6

MEAN VALUES IN THE SPACE

AIR VELOCITY	= 0.1456 m/s
AIR TEMPERATURE	= 22.7719 deg.C
CONCENTRATION	= 444.8944 *E-6

COMFORT NUMBER FOR AIR QUALITY

MEAN CONTAMINANT PRODUCTION RATE	= 4.7000E-3 l/s
CONCENTRATION AT THE EXIT	= 0.5330E+3 *E-6
VENTILATION EFFECTIVENESS	= 0.2459E+3 %
AIR FLOW RATE FOR ONE OLF	= 0.3000E+2 l/s/olf
PERCENTAGE OF DISSATISFIED FOR IAQ	= 0.5452E+1 %
COMFORT NUMBER FOR AIR QUALITY	= 0.4511E+2

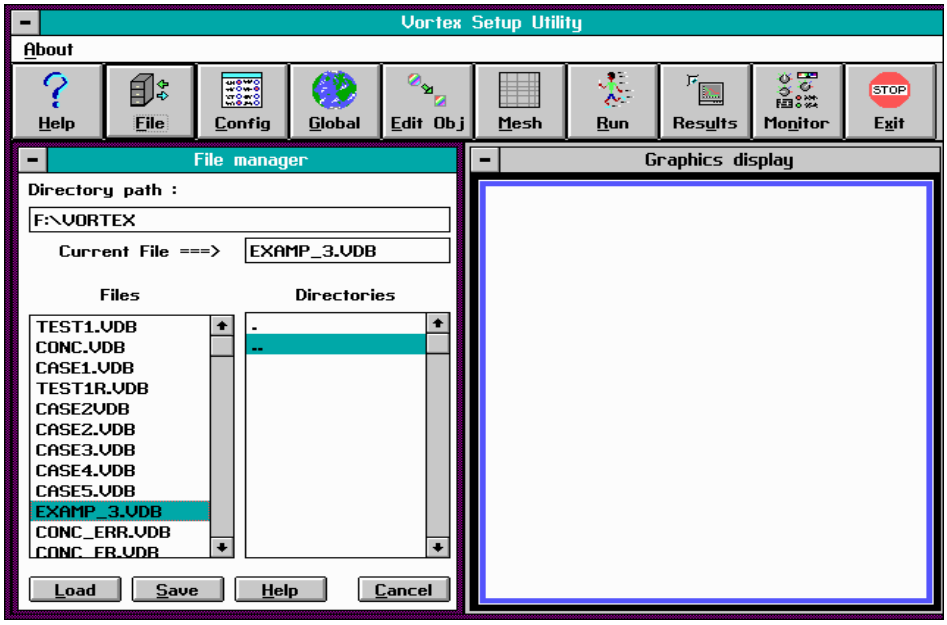


Figure 1: Vortex Setup Utility - File Manager

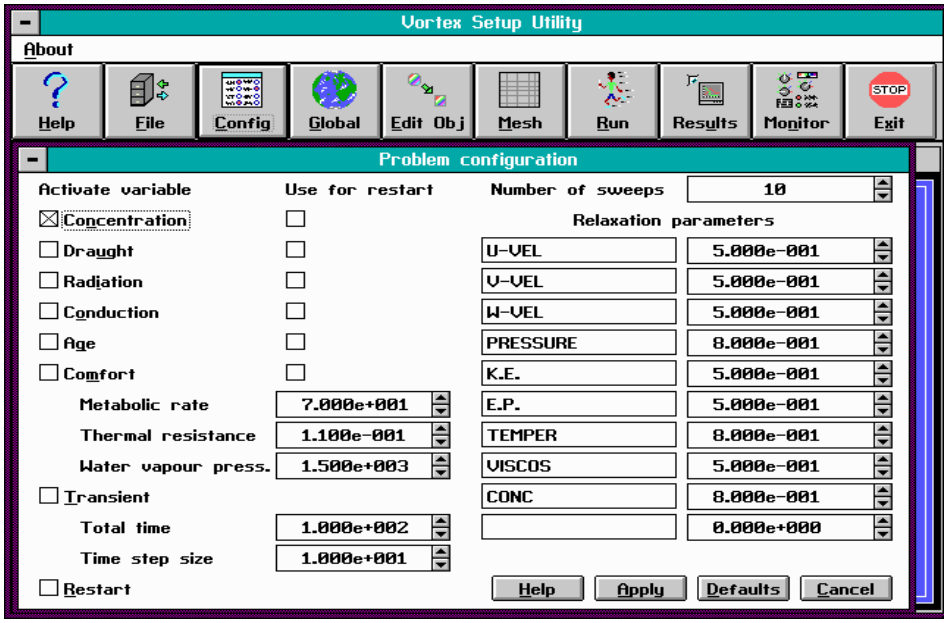


Figure 2: Vortex Setup Utility - Problem Configuration

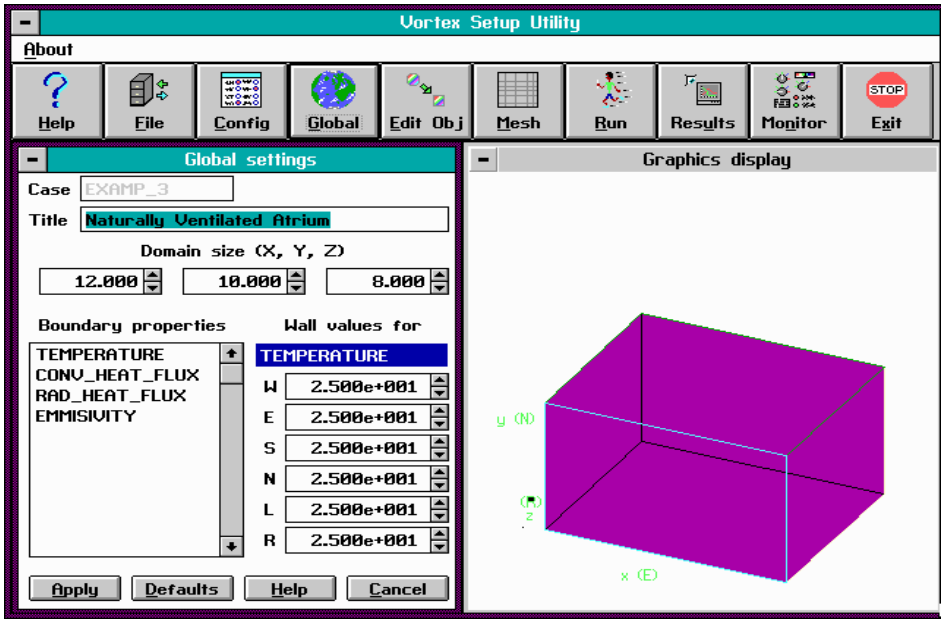


Figure 3: Vortex Setup Utility - Global Settings

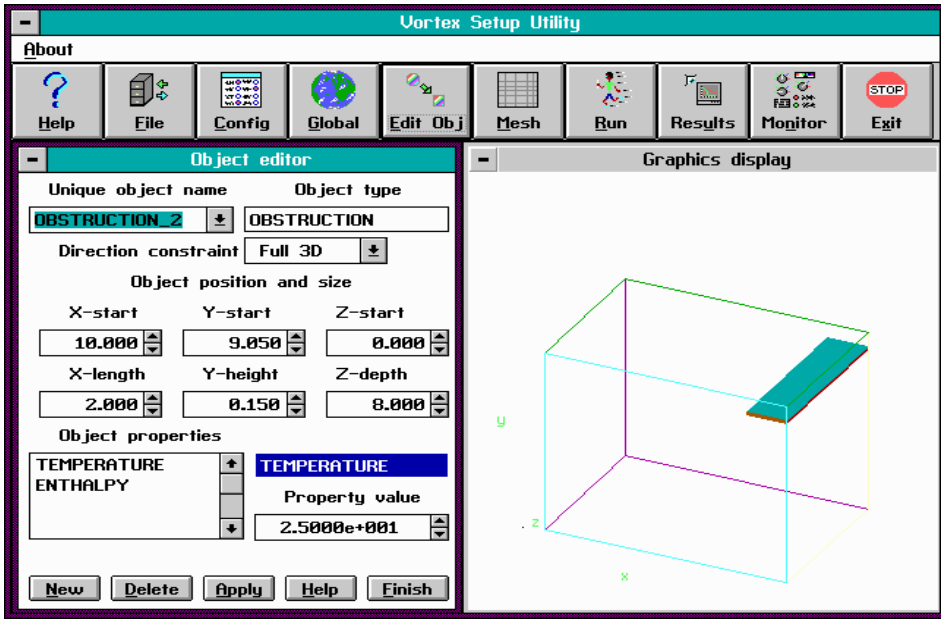


Figure 4: Vortex Setup Utility - Object Editor

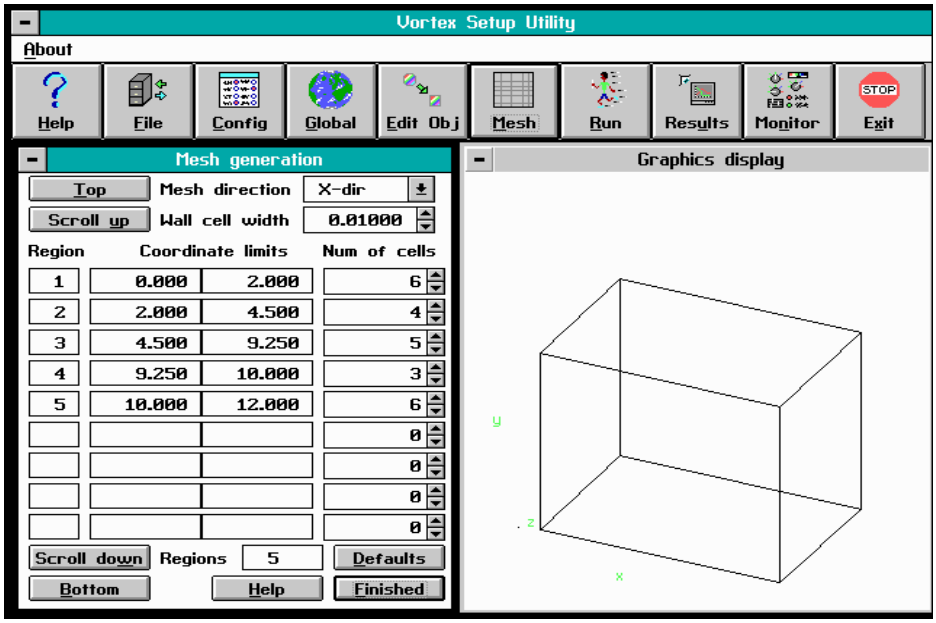


Figure 5: Vortex Setup Utility - Mesh Generation

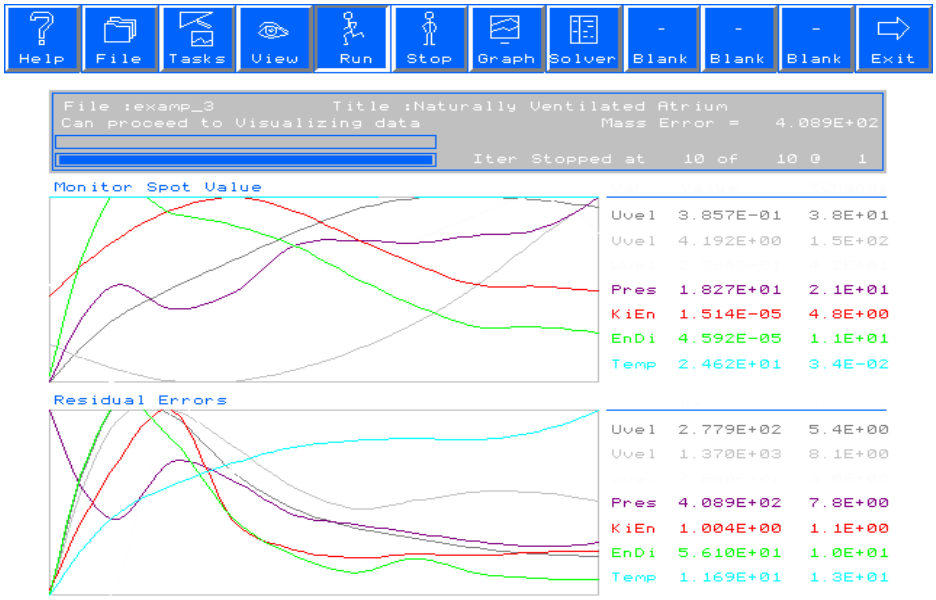


Figure 6: Vortex Setup Utility - Runtime Monitoring

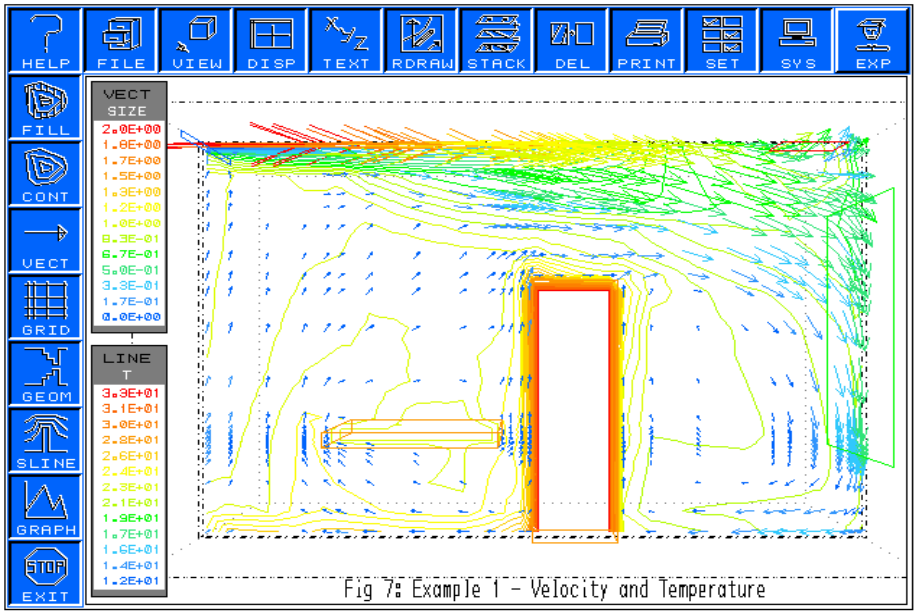


Figure 7: Example 1 - Velocity and Temperature

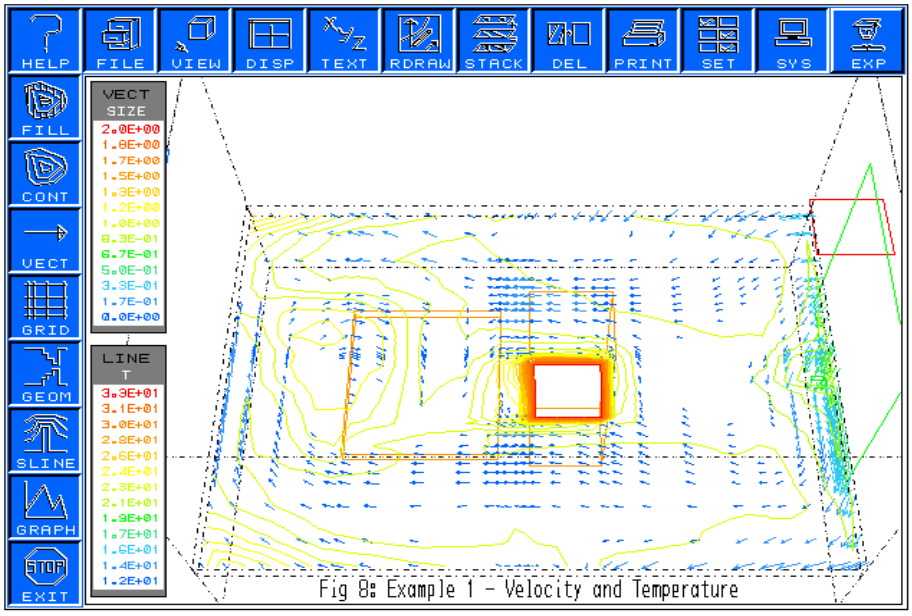


Figure 8: Example 1 - Velocity and Temperature

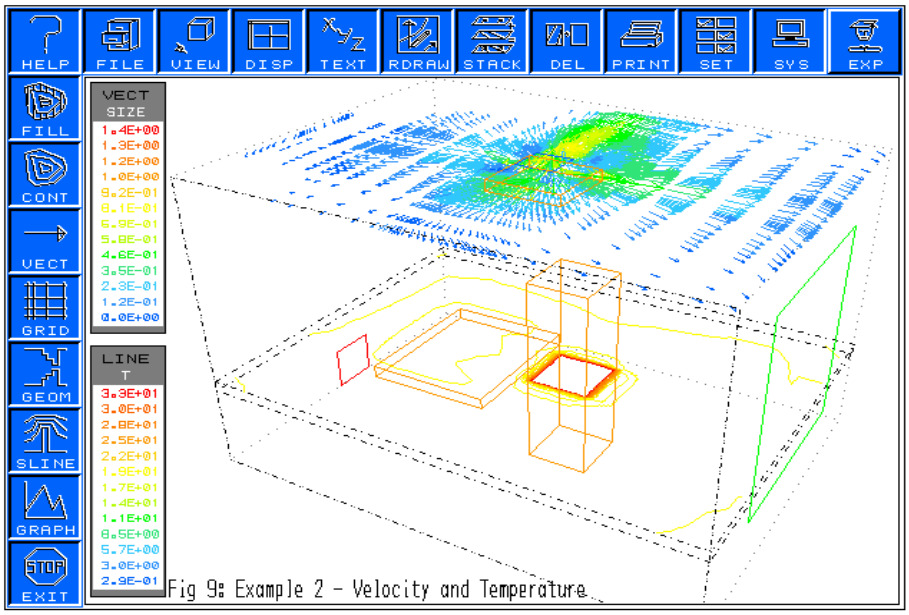


Figure 9: Example 2 - Velocity and temperature

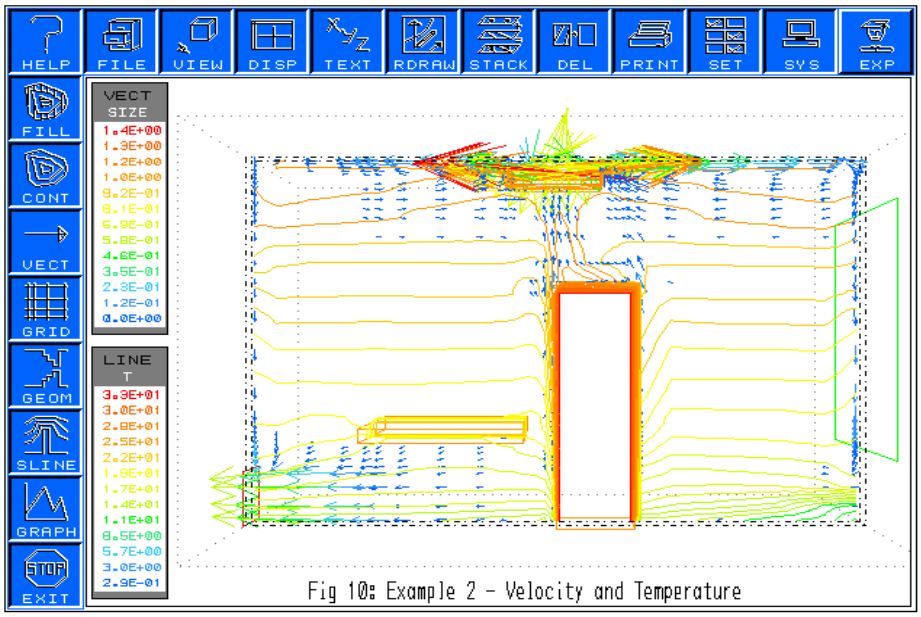


Figure 10: Example 2 - Velocity and Temperature

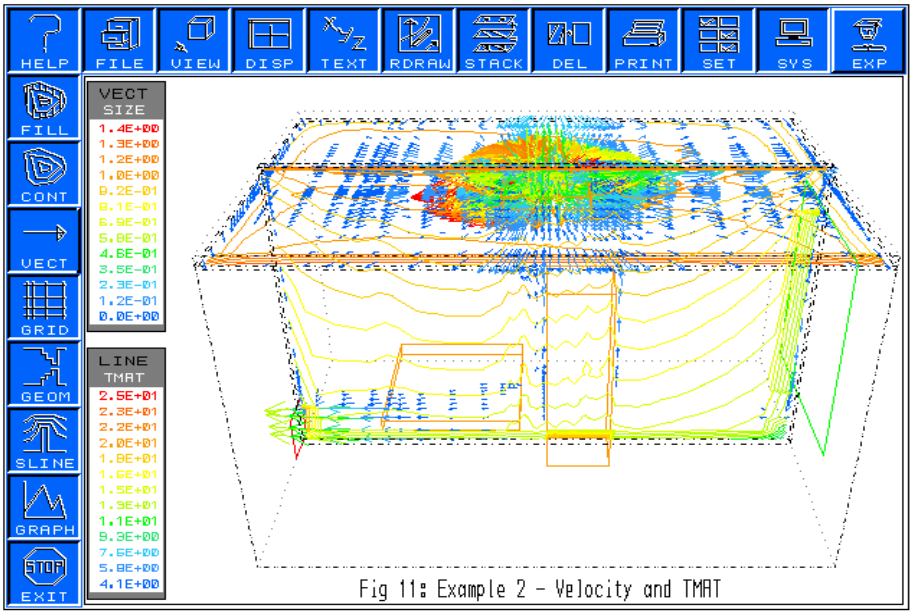


Figure 11: Example 2 - Velocity and TMRT

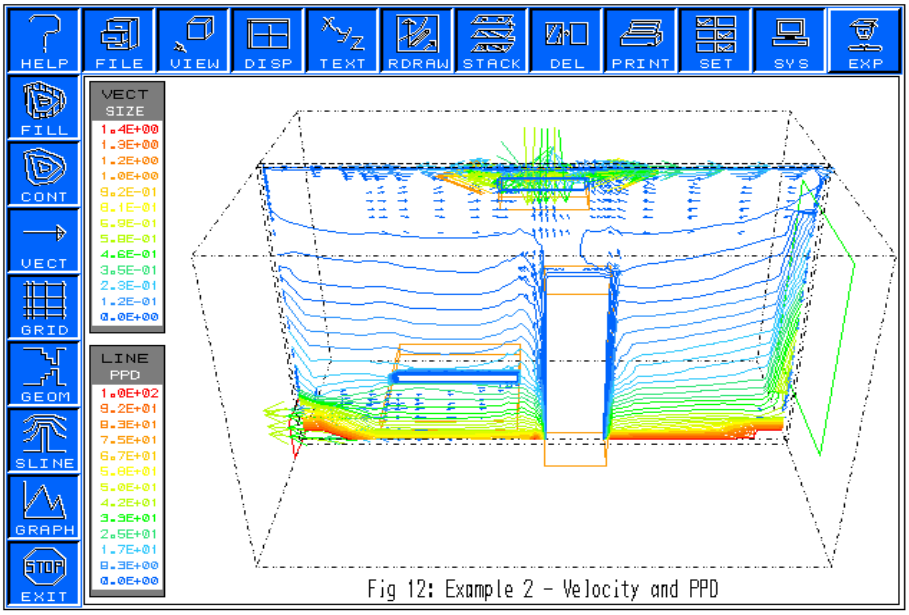


Figure 12: - Velocity and PPD

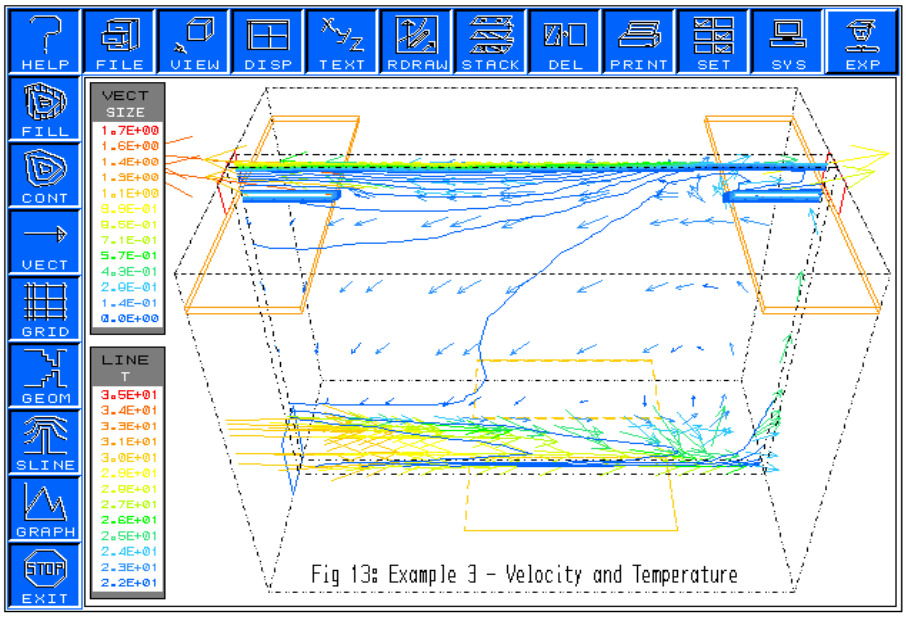


Figure 13: Example 3 - Velocity and Temperature

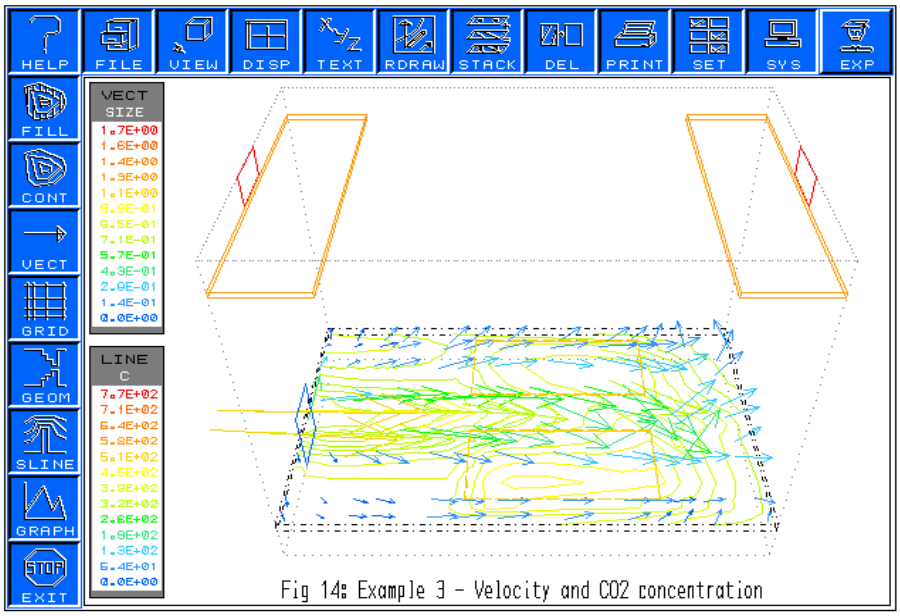


Figure 14: Example 3 - Velocity and CO2 concentration

9. INTRODUCTION TO FLOWVIS

There are often problems with existing visualisation systems. These include :-

·General purpose visualisation tools do not always treat fluid flow simulation data correctly. There are instances where data is misplaced or misrepresented.

·Post processing visualisation of fluid flow data is one of the most important aspects of simulation because the information and patterns in huge numerical data sets cannot be interpreted without accurate visualisation tools.

·Many visualisation systems have over-complicated user interfaces with so much in-appropriate functionality that users can be confused.

·Others provide cryptic command line interfaces that have intolerably high learning overheads. This is often accompanied by endless questions for the settings associated with a particular command.

·Where menu driven interfaces are provided these are often built onto command driven code. This gives a user interface that can be complex to control because the underlying development was command driven.

Why is ***FLOWVIS*** a better visualisation tool?

*·**FLOWVIS** was designed primarily as a post processing tool for visualisation of fluid dynamics data. The requirements were for intuitive user interaction and accurate representation of simulation data.*

*·**FLOWVIS** development is driven by criticism of existing visualisation packages. This has resulted in a system specifically for fluid dynamics researchers.*

*·The use of interactive interface techniques and intelligent default settings mean that **FLOWVIS** is both easy to use and has low learning overheads for users. Productivity for data analysis is enhanced by the interactive visual interface and the WYSIWYG view and data selection menu.*

·The development has initially been targeted specifically at PC platforms because of the increasing use of PCs for fluid dynamics simulation. The package is highly optimised to use the facilities of PCs for producing visualisations quickly and efficiently.

10. HARDWARE AND SOFTWARE REQUIREMENTS

FLOWVIS will run on IBM-PCs and most 100% compatible computers with the following hardware and software minimum specifications :-

· 80486-SX processor (or higher specification processor).

· A colour VGA card for displaying a 640x480 pixels in 16 colours.

· Colour VGA monitor capable of displaying 640x480 pixels in 16 colours.

· 32 Mbytes or more of main system RAM.

· 10 Mbytes of available Hard-Disk space.

· Windows 95 and above (including Windows 98 or Windows NT)

· 3 button mouse and driver software (pre-installed before running ***FLOWVIS***).

11. OVERVIEW OF ***FLOWVIS***

FLOWVIS is a post processing visualisation system for use with structured mesh fluid dynamics data. The system is fully menu driven using mouse or cursor selection. The main menu consists of 20 icons that occupy the left and top edge borders of the screen. Generally the selection of an icon will either initiate some action or invoke a sub-menu, which will pop-up over the central portion of the screen.

Sub-menus generally have a common top row of buttons as [HELP], [EXIT], [RESET] and ([DONE] or [DRAW]) from left to right. The rest of the menu will contain buttons, informative text, displayed data and status fields and some menus have graphical display regions that will contain images. Most of the menus are treated as forms that have a number of default values. The user can change these values by pressing (selecting-on) the associated buttons. When a menu contains the correct information the [DONE] (or [DRAW] on display-command menus) button will initiate the action required for that menu.

FLOWVIS only allows the selection of buttons. The data fields cannot be altered by selection on the data field itself. This strategy was adopted to avoid the confusion found with some other user interfaces where users are not sure which menu items they can and cannot select. Often the selection of an option will be followed by an automatic shift of the cursor to the next most logical option. For example the FILE MANAGER menu will shift the cursor from a button that selects a file to the [OPEN] file button ready for that action. Another example of the automation in ***FLOWVIS*** is the fact that the VIEW SELECTION menu is always presented after the FILE MANAGER menu has been used. This is useful as the next operation, after loading a new data-set, is usually to select the data region to be viewed and the view orientation.

FLOWVIS uses a data-base of files for visualisation. There will generally be more files in the data-base than there were input files. This is necessary because most CFD output files are designed to store data compactly but are not very fast to access. The ***FLOWVIS*** data-base is optimised for fast access and contains many calculated values which are not present in the original input files. The data-base creation routine can be accessed from within the ***FLOWVIS*** file manager and there is a command line version. The data-base files for ***FLOWVIS*** are easily identified by the three character extension which starts with an “F”. For example the file with extension “.FHD” is the header information for this data-set. One data-set uses the same eight character name for all associated files.

When [OPEN] is selected, for a data-base, in the FILE MANAGER all the data-base files are connected for reading. At this stage ***FLOWVIS*** is ready for your visualisation commands.

The main menu consists of icons on the left and top edge of the screen. The left edge has display commands e.g. VECTORS, CONTOURS and GRID etc. The top border has options for system configuration, file management, orienting the view and textual notation etc. The options have internal checks to prevent meaningless actions so that time wastage is kept to a minimum. An example of this is that menus are inactive until a data-set is connected. Also the PRINT option is disabled until there are some saved commands to redraw.

One of the main features of ***FLOWVIS*** is the command stack that saves commands for subsequent replay. As a drawing command is completed the command and associated settings are saved to the stack. The stack also maintains the view orientation in up to four windows. Automatic checking prevents commands that would produce meaningless displays. Examples of these checks include:-

1.Only one data-set can be used in one window.

2.There can only be one active view per window.

3.Graphs cannot be mixed with other commands in one window except text.

If a possible conflict occurs ***FLOWVIS*** will prompt you for keeping the existing display or updating the display. It must be stressed that updating the display can mean that previous commands are purged depending on the type of conflict.

The fact that ***FLOWVIS*** maintains a view for each display window means that up to four differently oriented views can be shown on screen simultaneously. The view menu can be used in conjunction with the redraw option to change the orientation of display elements in one window. If the currently set view is different from the view in a target window, as a command is about to be drawn, ***FLOWVIS*** will ask if the view should be updated or the command displayed using the previous view. If the display is updated to the new view orientation then the previous commands will be replayed to create the composite display in the new orientation.

There are a number of ways of saving images from ***FLOWVIS***. Standard hard-copy to file is provided in HP-GL, PostScript and Raster formats. These files have to be created by redrawing all the commands in the display. An alternative is the PCX screen dump facility which captures the entire screen and saves it as a PCX file. Finally there is a facility for bitmap image saving and loading which can be used for screen-shot slide shows. These bitmap images are not printable and are not particularly compact but have the advantage of fast display.

The displays created by ***FLOWVIS*** are mostly created on a plane by plane basis. The exceptions are streamlines and contour nets which are continuous through the volume of the domain. ***FLOWVIS*** warns you about view orientations that present the planes edge on rather than flat on the screen. ***FLOWVIS*** will also attempt to draw certain commands in such a way that minimal overwrite occurs.

This is achieved by trying to create the display drawing from deepest to nearest screen depth coordinate. This technique is by no means infallible but generally produces acceptable displays.

Finally the drawing commands can be broken into and stopped. There are two different ways of stopping drawing. The first is to use the [ESC] key or the right hand mouse button. This will cancel the current operation and will not save the command to the redraw stack. The second break in action uses the numeric keypad [ENTER] key or the middle mouse button. Although this terminates the current drawing operation it will still save the command to the redraw stack for subsequent redrawing.

12. GETTING STARTED WITH *FLOWVIS*

If you want to use a mouse with ***FLOWVIS*** then your mouse software driver should be loaded into memory before you attempt to run ***FLOWVIS***. The ***FLOWVIS*** executables MUST be invoked with the batch script files that are present in the directory called ***FLOWVIS***. These scripts are tailored to your particular installation and set some environment variables for use by ***FLOWVIS***.

If you do not have any prepared ***FLOWVIS*** data-sets then you must create some before you can visualise any data. The ***FLOWVIS*** data-sets can be created from the command line using the FILTER utility or using the [CREATE] option from within the ***FLOWVIS*** FILE MANAGER menu. See the section in this manual on "CREATING A ***FLOWVIS*** DATA-SET".

The ***FLOWVIS*** script file optionally takes a data-base case name argument. This feature allows ***FLOWVIS*** to immediately load the named data-set. If no case is specified or the case was not found then an introduction screen will be displayed. At this point you can use a mouse or keyboard selection to confirm any options. Mouse selection is simply a matter of highlighting the button you wish to operate and then selecting with the left mouse button. The cursor keys can also be used for changing the highlighted button and the [SPACE] or [ENTER] keys will operate the button.

If no file has been pre-loaded (see above) then the first menu automatically presented is the FILE MANAGER menu. This allows you to change directory (if required) and to open a named data-set for visualisation. If you wish to change drive you must use the button marked [PATH]. This feature is a little awkward on DOS systems but has the advantage of maintaining compatibility with unix workstations where there are no named drives to access. The main facility provided by the FILE MANAGER is to [OPEN] a new ***FLOWVIS*** data-set for visualisation.

Once a file has been loaded the FILE MANAGER menu automatically dismisses and the DISPLAY MENU will appear. This menu provides options for changing the number of windows used in the display. You can experiment with any of the features here but the default configuration can be used simply by pressing the [DONE] button.

The VIEW SELECTION menu is also automatically presented. You should use the view selector to choose the extent of the data you want to be used for subsequent drawing commands. The VIEW SELECTION menu should also be used to set the orientation used to draw any of the display commands. The menu has a graphical area that provides comprehensive and interactive visual feedback of the set view orientation and the extent of the data selected. When you have set the required view and data-extent you can select [DONE] to return to the main menu. At this point ***FLOWVIS*** is ready and waiting for your display commands.

At the main menu the icons at the left side and top of the screen can be selected by mouse or cursor keys and [ENTER] to confirm. Most of the icons pop-up a menu over the centre of the screen. The options for displays to be created are on the left of the screen. Although many of the display menus look quite complicated the use of intelligent defaults means that very few options actually need to be changed. Generally only the variable used or type of display will have to be changed for visualisation

13. MAIN MENU OPTIONS

The main menu options are described in general overview of the functionality. If the main menu option opens a sub-menu then the reference material for each of the buttons in the sub-menu is described.

Sub-menu descriptions are generally supported by a graphical image that shows the screen representation of the open menu. Where collections of buttons are associated with a single data field their functionality is described as a group.

There are essentially three types of button in sub-menus. The first type change the value of a parameter that is displayed in a data display field. This may involve the addition or subtraction of a value or stepping through a set of options. These buttons are often labelled as [<], [>], [-] or [+]. Occasionally it is necessary for a button to change the parameters displayed in a number of the display fields.

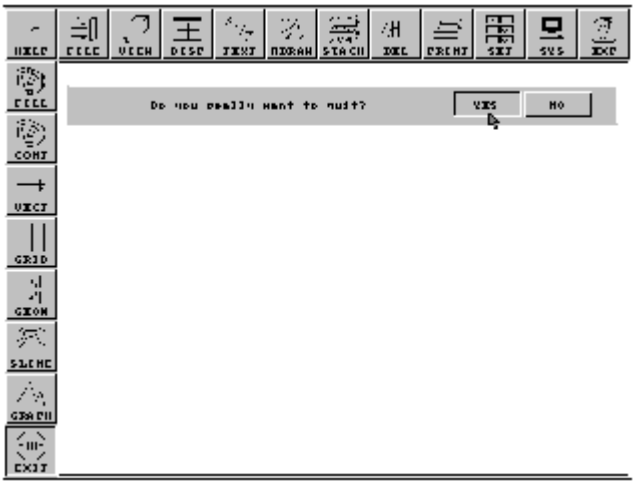
Another type of button opens a textual input line where characters or numbers can be entered. These buttons often have the label [SET] to indicate the manual setting of a value.

Finally there are "do-it" buttons that immediately initiate some action. The [DRAW] and [EXIT] options are examples of buttons which immediately perform some action.

Most of the sub-menus are form menus. The parameters can be modified until all of the selections are as desired then a single button selection initiates the functionality of the menu using the selections in the form. Almost all of the sub-menu parameters are defaulted to sensible values so there are generally only minimal changes required to draw a picture element.

EXIT

Exit overview



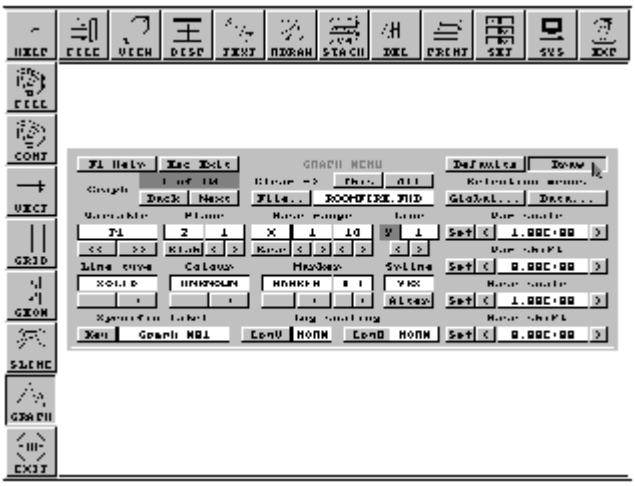
Exit allows you to quit from ***FLOWVIS*** and return to the operating system. The user is prompted for confirmation of selecting quit.

If the exit is confirmed ***FLOWVIS*** will check the redraw stack for drawing commands. If there are commands in the stack then you will be asked if you want save the stack as a file.

Just before ***FLOWVIS*** *terminates any temporary files will be deleted and all open files will be closed. Finally the normal text screen is restored.*

GRAPH

Graph overview



The graph option allows up to ten graphs to be specified for one target window. These graphs can be for any cell line within the data-set. The choice of cell line (and the range of cells) is achieved by selecting the data plane that contains the line of cells, the range of cells in the cell line and the other line index which identifies one particular line of cells from the plane. The limits of cells along the chosen line are termed the "BASE RANGE", The coordinates of the cells in the specified range are plotting along the horizontal (or base) axis of the graph. Any one of the variables from the data-base may be used as the vertical axis.

Most of the GRAPH MENU contains options associated with one particular graph-line selection. The [BACK] and [NEXT] options can be used to access any one of 10 line-graph settings.

Options are provided for markers, drawing a smooth spline of the data points, changing colours, changing line type and for scaling or shifting a particular graph. There is also access to a visual data selection menu which indicates a line of cells. This can be used to interactively choose the graphing location with visual feedback confirmation. The menu used for data selection is very similar to the VIEW SELECTION menu except that the view orientation, in this menu, is only for your convenience and is not used with any other displays.

When you select [DRAW] to display the specified graphs ***FLOWVIS*** automatically scales all the graphs to fit within the target window. The graphs are saved to the redraw stack and when graph is selected again (for a particular window) all of the specifications will be available for modification.

FLOWVIS *allows you to access other data-sets simultaneously for graphing. The [FILE] button actually brings up the FILE MANAGER menu in the correct mode for opening a data-base. The active data-base name, for each graph, is displayed next to the [FILE] button.*

Graph options

Use the [HELP] button to enter help mode for the GRAPH sub-menu. The selection of any button in help mode will bring up help for that button or group of buttons. The help mode is exited by selecting on the [HELP] button again. The [F1] key will also perform the same action but will present help for the last highlighted button.

The [EXIT] option allows the GRAPH sub-menu to be exited without any changes taking effect. This action can also be performed with the [ESC] key or the right-hand mouse button. No attempts are made to do displays and changes to parameters will be lost.

The [DEFAULTS] option resets the default parameters in the GRAPH menu. This provides a quick way of returning to sensible values.

The [DRAW] option takes all of the GRAPH specifications and calls the graph display routine. In displays with more than one window ***FLOWVIS*** will draw the graphs in the same window as was chosen when the main menu GRAPH option was selected. All of the specified graphs are re-drawn from scratch. If there is any conflict of the graph commands with existing display commands then you will be asked if you require the existing commands to be kept or discarded and replaced by the graphs.

The [BACK] and [NEXT] Graph menu options are provided to step through the individual graph specifications. The actual graph specification being modified is indicated by the numbers displayed above these buttons. Each graph uses the whole menu as a form that specifies only that one graph. The other graph specifications are then accessed by stepping forwards and backwards through the forms. This will often be accompanied by a number of changes in the data display fields.

The Graph [FILE..] option allows other data-sets to be accessed for graphing. These are opened in the file manager menu. The filename displayed to the right of the file button indicates the data-set from which this particular graph will be drawn. The selection of variable and the cell extent will be set appropriately for this data-set.

The clear [THIS] and [ALL] options are used to clear one or all of the graph specifications respectively. This is accomplished by setting the variable for one or all graphs to "NULL". The graphs with "NULL" as the variable are ignored when draw is selected.

The [GLOBAL...] option will be available soon to provide facilities for global graph settings. These facilities will include access to the graph title and labelling of axes, graph graticules, axes settings and presentation styles. This is to be implemented as another menu which will exit back to the graph specification menu.

The [DATA...] option opens a visual menu provided for interactive selection of the line of cells to be used for graphing. The data selection menu is very similar to the main view menu but is restricted to selecting only one line of cells. The menu does feature the view orientation options of the main view menu but any changes made in this menu are not used in subsequent displays. The data selection menu exists back to this graph specification menu and updates the cell line extent for the current graph element.

The variable selection [<<] and [>>] options allow the variable used for the current graph element to be selected. These buttons scroll through the entire list of variables that are available in the current data-base. The selected variable is indicated by name above the buttons. A "NULL" in the variable name field means that the current graph element is inactive as no variable has been chosen.

The [SLAB], [<] and [>] options allow the graphing data plane to be selected. This is one stage of the selection of a line of cells. The plane or slab orientations is chosen with the [SLAB] button as an X, Y or Z plane and the [<] and [>] buttons step through the available planes by index. The actual plane currently selected is indicated by the display fields above these buttons. The chosen plane is the plane that the line of cells will be taken from for graph display for this particular graph element. The Base range and Line options are used to select the exact line of cells used for this graph element. Changing the plane orientation automatically changes the base range and line orientations because they must all be mutually exclusive directions. Data-sets which are 2-Dimensional will not permit the plane orientation to be altered.

The [BASE] orientation and base limits [<][>], [<][>] options are provided to choose the orientation and extent of the line of cells used for this graph element. The direction of the line of cells and the end limits are displayed above these buttons. Once a plane orientation has been

selected there are only two possibilities for the base-range direction. The exact cell line to be used is not specified fully until the third coordinate direction index is selected.

The cell line [<] and [>] options allow the third coordinate index for this graph element to be selected. The direction and index are indicated above the buttons in the display fields. The direction cannot be changed because the plane and base directions have already been selected leaving the third coordinate direction for this option.

The variable scale [SET], [<] and [>] options allow the scale of the variable to be changed for this graph element. By default the scale value is 1.0 which does not scale the values of the variable. The scale value can be set explicitly using the [SET] option or stepped to using the [<] and [>] buttons. The scale is applied by multiplying the value in each graph cell by the scale value. Negative numbers are allowed.

The variable shift [SET], [<] and [>] options allow the variable axis to be shifted. By default this shift value is set to 0.0 implying no shift. Positive values have the effect of increasing all of the values of this variable whilst negative values decrease all the variable values. The [SET] button allows explicit selection of a shift value whereas the [<] and [>] buttons step incrementally.

The base scale [SET], [<] and [>] options are used to scale the coordinate axis of the current graph element. The scaling is applied by multiplying every coordinate point in the graph by the scale value. By default this value is set to 1.0 implying no scaling. The [SET] option can be used to set the coordinate base scale explicitly whilst the [<] and [>] buttons step incrementally to a value.

The base shift [SET], [<] and [>] options allow all of the points in this graph element to be coordinate shifted by the shift value set here.

A positive value increases the coordinate values whilst a negative value decreases the coordinate values. The [SET] button allows the shift value to be typed in explicitly whilst the [<] and [>] buttons step incrementally to a value. The line type [-] and [+] options allow the line style for this graph element to be changed from the default. There are options for dotted, dashed, dot-dashed and no-lines. The graph element is drawn using the indicated line type. When “NO-LINES” is selected the markers will be used to show the graph location.

The colour [-] and [+] options allow the colour of each graph element to be selected. The colours scroll through a list of named colours. The colour used for this graph element is the one named above the buttons in the display field. The marker type [-][+] and marker character [-][+] options allow the selection of a particular type of marker to be used for this graph element. The marker types are marker, letter and number or none and there are ten choices for the character used. The markers indicate the locations of the actual data point (x,y) pairs on the graph element. Also the specific key label has the appropriate marker type for referencing the correct graph element.

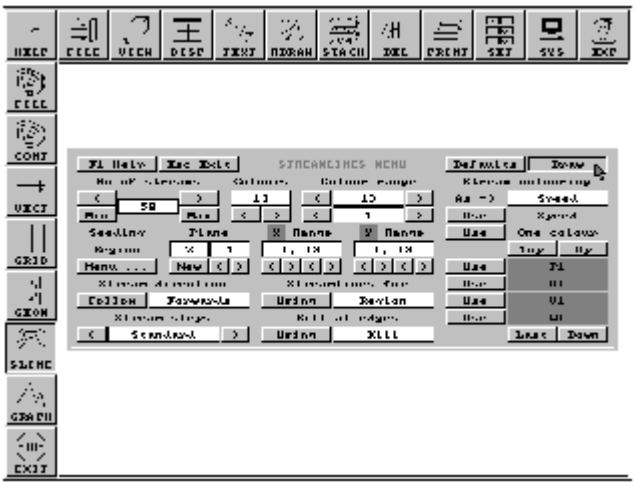
The spline [ALTER] option allows a graph element to be drawn with a smooth spline. When spline is used there may be overshoots where there is a very sudden change in gradient but for the most part spline produces a smooth curve through the data points.

The [KEY] label facility allows you to input a limited amount of text to reference individual graph elements. Currently 10 characters are allowed. These form a reference key drawn to the side of the graph. Markers (if used) identify the specific keys to graph elements but also the keys are coloured similarly to the appropriate graph.

The [LogV] and [LogB] options allow the graph variable and / or base axis to be logged. This will only be appropriate if all of the values are positive otherwise the request is ignored. It should be stressed that mixing log and non-log graphs together may have the effect of minimising one of the graphs almost to obscurity because of the automatic scaling and the ranges of values.

STREAMLINES

Streamline overview



The formal definition of streamlines states that streamlines are, at all points, tangential to the local velocity field. ***FLOWVIS*** allows a 2-Dimensional plate region to be seeded with streamlines which can then be followed forwards or backwards.

The streamlines are, by default, coloured by the local velocity magnitude but can also be coloured by any data-base variable or in a single colour.

The number of streams seeded can be selected from 1 to 250. These are scattered regularly on the seed region. The accuracy of the streamlines can be selected using the streamline steps parameter. This indicates how many steps of calculation should be used to get across a cell. The larger this value the more accurate the streamlines (and the longer the display takes). The plate region for streamline seeding can be selected from indicated indices or from a plate (or slab) data selection menu. The SLAB SELECTION menu is very similar to the VIEW SELECTION menu but the orientation within the slab menu is not used in any other displays.

Streamlines, particularly ones which get caught in recirculations, may not terminate so ***FLOWVIS*** *imposes a number of termination conditions. After a set number of stream following steps the user is prompted for continuing to following the streamlines.*

FLOWVIS will not follow a particular streamline if it exits the domain or if it appears to stop. The second condition is necessary because streamlines are independent of time and the scaling of very small local velocities could produce spurious results.

When the streamlines are drawn some may appear to advance faster than others. There is NO physical significance in this and it is merely a feature of the streamline tracking algorithm used.

Streamline options

Use the [HELP] button to enter help mode for the STREAMLINES sub-menu. The selection of any button in help mode will bring up help for that button or group of buttons. The help mode is exited by selecting on the [HELP] button again. The [F1] key will also perform the same action but will present help for the last highlighted button.

The [EXIT] option allows the STREAMLINES sub-menu to be exited without any changes taking effect. This action can also be performed with the [ESC] key or the right-hand mouse button. No attempts are made to do displays and changes to parameters will be lost.

The [DEFAULTS] option resets the default parameters in the STREAMLINES menu. This provides a quick method of returning to sensible values after experimentation.

The [DRAW] option takes all of the STREAMLINE settings and calls the display routine. In displays with more than one window ***FLOWVIS*** will prompt for the target window in which to display STREAMLINES. Streamline following will proceed until all of the streams have been terminated or a number of cells have been stepped through. If there are still streams active when the preset number of steps have been reached then ***FLOWVIS*** will prompt for streamlines to continue for a similar number of steps.

The [<][>], [Min] and [Max] options allow the number of seeding points for Streamlines to be selected. The specified number of seeds are then placed regularly in the selected seed region. The seed region is a given 2-Dimensional plate within the domain.

The colours [<] and [>] options allow the number of colours used for following Streamlines to be set. The default is the maximum number of colours available. This option only has effect between the specified range of colours. It is used for limiting the number of colours used from a potentially large range of colours.

The Colour Range upper limit [<][>] and lower limit [<][>] options allow the actual colour range used to be selected. The colours range from red as the lowest number to blue which is the highest. The range can be reduced, reversed or switched to white only by using the middle mouse key.

The seeding region [MENU...] option brings up a visual menu, similar to the main menu view option, for selecting a streamline seeding 2-Dimensional plate. This plate can also be selected using options in this menu. The plate (or slab) forms the region over which the streamline seeds are placed.

The Streamline plane [NEW], [<] and [>] options allow the seeding slab to be changed to an X-, Y- or a Z-plate and the number of the slab to be chosen. The streams will be seeded regularly over the extent of this slab. For 2-Dimensional domains these options have no effect.

The seeding slab ranges [<][>][<][>] options can be used to reduce the extent of the seeding region down from the whole plane to a sub-region. The extent is specified by cell indices and the data-base directions are displayed above the ranges.

The Streamline [FOLLOW] option allows the direction in which the streamlines are followed to be specified. When the streamlines are followed forwards the velocities are used as they are in the data-base. Following backwards uses the negated velocity components found in the data-base.

The streamline [USING] option allows streams to be tracked in the whole domain or optionally only in the current view selected region. If a seed starts outside of the region (or leaves the region) it will not be followed any further.

The Stream steps [<] and [>] options allow the streamline refinement to be specified. This integer value determines how accurately (and fast) the Streamlines are followed. The larger the value the more steps are used to follow the Streams. More accuracy does mean that the stream following will be slower.

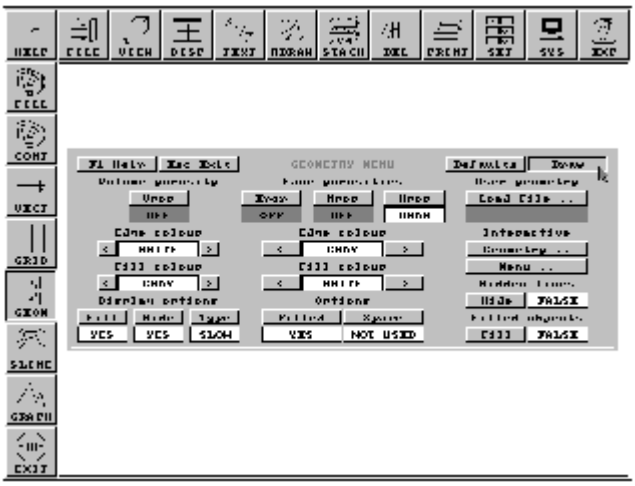
The kill at edges option has not yet been implemented.

The stream colouring [USE] options allow the type of stream coloration to be selected. The choices are for local resultant speed, single colour or using a data-base variable for coloration. The colouring used is indicated just below the [DRAW] button.

The scroll [TOP], [LAST], [UP] and [DOWN] options allow other portions of the variable list to be displayed. This is necessary where the full variable list is larger than the visible portion of the list. The [USE] buttons allow the adjacent variable names to be selected.

GEOMETRY

Geometry overview



Geometry is supported in two distinct ways. The first is by porosity specified geometry given as field (or cell) values in the data set. This can be cell porosity (Volume PORosity) or face specified porosities for high-x (East PORosity), high-y (North PORosity) or high-z (High PORosity). These geometrical features are displayed as filled cells or tiles respectively. The fill and edge colours can be selected for these volume or face porosities. By default the volume porosity geometry routine uses an algorithm which sorts the cells into furthest to nearest order and displays them using hidden line techniques. Whilst this takes a little while to sort the cells in order it does produce pleasing displays because the "block" objects have a solid appearance.

The other form of geometry supported is by user defined features. These are entered as commands which display boxes, arcs, lines, poly-lines and polygons etc. These commands (and their associated parameters) can also be interpreted from a geometry file.

Two other commands (OPEN and CLOSE) allow interactively specified geometry to be saved to a file. This is necessary if the geometry is to be saved to the redraw stack. Closing the geometry file causes the commands to be available for redrawing. The geometry menu also has an option to bring up the file manager for choosing an existing geometry file for replay.

FLOWVIS does not check the type of file that is requested for geometry so the user must be sure that the file does contain geometry commands.

Geometry commands

The commands and parameters used for specifying user defined geometry are:-

OPEN <filename>

Which opens a named new file for saving geometry commands or opens a file for replay.

CLOSE <filename>

Which closes the named or currently open file and ensures that it will be available for redrawing.

ARC line colour circ

+ x1 y1 z1

+ x2 y2 z2

+ x3 y3 z3

Draws an arc using three coordinate points on the arc. If circ is 1 then a circle will be draw rather than just an arc for a value of O.

the command is typed over four lines using the + to indicate successive lines of input.

BOX x1 y1 z1 x2 y2 z2

Draws a box with diagonally opposite corner coordinates at (x1,y1,z1) and (x2,y2,z2).

LINE x1 y1 z1 x2 y2 z2

Draws a line from the coordinate point (x1,y1,z1) to point (x2,y2,z2).

PLINE x1 y1 z1 (line) (colour)

+ x2 y2 z2

+ ...

+ xn yn zn

Draws a poly line for the given coordinate specified points. The continuation character + is used to add points over subsequent lines of input. At least 2 points must be given in which case a single line is displayed.

PGON x1 y1 z1 (line) (fill)

+ x2 y2 z2

+ . . .

+ xn yn zn

Draws a polygon for the given coordinate points. The continuation character + is used to add the next line segment. The polygon will only be displayed after the last line of input. End of input is indicated when a blank line is entered.

GRID OUT orient plane OR

GRID OUT orient plane dir1 lim1 lim2 dir2 lim3 lim4

Draws the grid outline for the given plane or the specified region. The ORIENTation, DIRection_1 and DIRection_2 are one of "X", "Y" or "Z" and must be exclusive so that each appears only once. The PLANE-number and LIMits are cell index numbers for the indicated directions. For example:-

GRID OUT X 3 Y 1 15 Z 2 4

will display the grid outline of plane X = 3 for the Y and Z limits given.

EXIT (or \)

Quits from interactive geometry specification.

HELP (or **?**)

Brings up on-line help.

HELP <Command_name>

Brings up help about a specific command.

Comments can start with **#** or **REM** and are ignored by the interpreter.

N.B. Parameters in parenthesis (option) are optional and names in angle brackets <name> should be replaced by the required name for that action.

Use the [HELP] button to enter help mode for the GEOMETRY sub-menu. The selection of any button in help mode will bring up help for that button or group of buttons. The help mode is exited by selecting on the [HELP] button again. The [F1] key will also perform the same action but will present help for the last highlighted button.

The [EXIT] option allows the GEOMETRY sub-menu to be exited without any changes taking effect. This action can also be performed with the [ESC] key or the right-hand mouse button. No attempts are made to do displays and changes to parameters will be lost.

The [DEFAULTS] option resets the default parameters in the GEOMETRY menu. This provides a quick method of returning to sensible values after experimentation.

The [DRAW] option takes the GEOMETRY settings and calls the display routine. In displays with more than one window ***FLOWVIS*** will prompt for the target window in which to display GEOMETRY. The interactive GEOMETRY option displays the user-defined geometry and exits back to the main menu automatically.

The volume porosity [VPOR] option allows volume porosity to be used (if available) for displaying blocked cells. If no volume porosities are present in the data-set then this option will not be available. The style of displaying cells is set using the options below [VPOR].

The face porosity [EPOR], [NPOR] and [HPOR] options allow blocked faces (defined by porosities) to be displayed. If there are no face porosities defined in the current data-set then these options cannot be activated. The style of display for the blocked faces can be selected using the options below the face porosity buttons.

The [LOAD FILE..] option allows the user to load and display the geometry commands from an ascii geometry file. This option brings up the FILE MANAGER menu to return a file name. The user must be sure that the selected geometry file does indeed contain geometry commands because ***FLOWVIS*** does not check the type of file. The results could be unpredictable if a binary file were opened and interpreted.

The edge colour [<] and [>] options for volume porosity cells allow the cell bounding edges to be displayed in a different colour to the faces. This option is used for the cell edges regardless of the status or colour of the faces.

The edge colour [<] and [>] options for face porosities allow the face bounding edge colour to be specified. The edges will be displayed in this colour regardless of the settings of face fill colour or face fill status.

The interactive [GEOMETRY..] option allows the user to enter geometry commands into an interpreter for the display of user-defined objects in the data-domain. The commands include coordinate specified boxes, arcs, polygons, poly-lines and cell region specified grid outlines. There are also open and close file commands that allow the typed commands to be saved to file for subsequent replay by the redraw facility.

The interactive geometry [MENU..] option is not currently available. This is being developed to provide interactive user defined geometry specification. These facilities will be available with future upgrades to the ***FLOWVIS*** system.

The fill colour [<] and [>] options for volume porosity blocked cells allows the colour of the cell faces to be selected. This option will be ignored if the volume porosity fill option is set to NO-fill.

The fill colour [<] and [>] options for face porosities allow the colour of the faces to be selected. This option will be ignored if the face porosity fill option is set to NO-fill.

The hidden line [HIDE] option is currently under development to allow the display of user defined geometry objects using hidden line techniques. This option will be ignored by this version of ***FLOWVIS*** but will be supported in future upgrades.

The volume porosity [FILL] option allows the faces of blocked cells to have solid filled faces in the colour specified by the fill colour option.

The volume porosity [HIDE] option allows cells to be displayed using hidden line techniques. These are slower but do prevent overwriting by cell faces that are deeper into the domain.

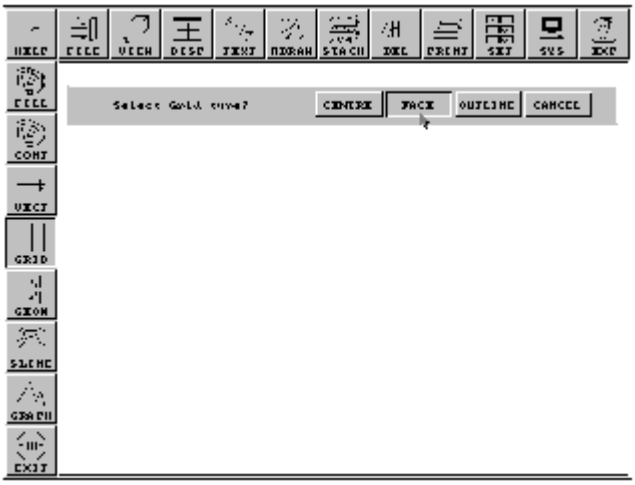
The volume porosity [TYPE] option specifies the speed with which the cells should be displayed. The fast option merely displays the cells in a far to near order so that near cells are not obscured. The slow option sorts the cells accurately by screen depth but does produce a more pleasing display.

The face porosity [FILLED] option allows blocked faces to be drawn using solid colour fills. The colour used will be the one specified in the face porosity fill colour option. The fill and colour options for faces have effect for all three cell face directions.

The user defined geometry [FILL] option is not currently available. When supported it will allow user defined objects to have filled faces where appropriate. This feature will be provided as an upgrade.

GRID

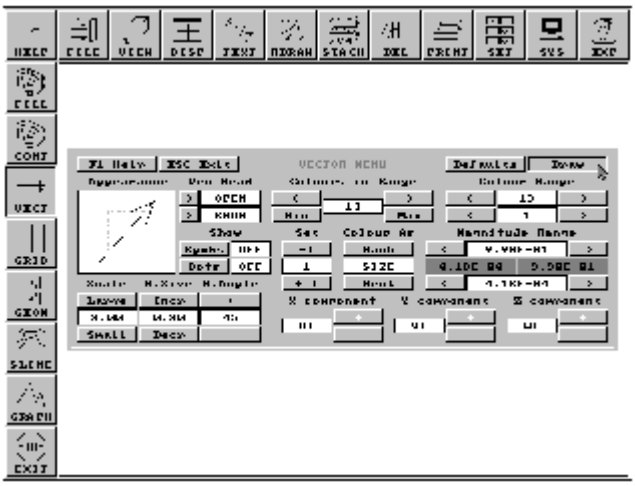
Grid overview



The grid display routine has options for cell centred, face centred or grid outlines. These are displayed at the mid-plane position. This is consistent with most of the other display routines which display properties at the cell centres. The two types of grids and grid outlines are only displayed for the currently selected data region.

VECTORS

Vectors overview



Vectors are plotted for orthogonal sets of vector components. The vector arrows are displayed at the centre of the cells within the domain. The values used are interpolated to the cell centres from the faces of cells. The vectors are automatically scaled to the average cell size. This helps to prevent vectors from overwriting portions of other adjacent vectors. There is a manual scaling utility to adjust the size of the vectors in situations where automatic scaling is unsatisfactory.

Vectors can be coloured by magnitude, any single variable from the data-base or single-coloured. The colouring ordering can be chosen to run from RED=greatest_value to BLUE=smallest_value or as RED=smallest_value to BLUE=greatest_value. The values concerned are the minimum and maximum values associated with the current domain or data region. See the SETTINGS MENU for information about the minimum and maximum scan options.

The vector sets are determined as the data-base is created and the selection of which vectors to display is then a simple matter of choosing the required set. The actual names of the components (in a set) are displayed at the bottom of the menu. The default set is always the cartesian component set for the first phase. Cylindrical polar data sets have their vector components resolved to cartesian componets at the data-base creation stage.

The vector menu contains a number of options to change the appearance of the vector arrows. The actual vector appearance is indicated by the graphical display area. This graphics box contains a representation of a vector arrow and a cell (a box drawn with a dotted line).

One other feature of the vector menu is the magnitude range selection. This indicates the current range of magnitudes in the data set for the currently selected region. The actual range displayed can be changed. This has the effect of discarding vectors which are outside of the specified range.

Vectors options

Use the [HELP] button to enter help mode for the VECTOR sub-menu. The selection of any button in help mode will bring up help for that button or group of buttons. The help mode is exited by selecting on the [HELP] button again. The [F1] key will also perform the same action but will present help for the last highlighted button.

The [EXIT] option allows the VECTOR sub-menu to be exited without any changes taking effect. This action can also be performed with the [ESC] key or the right-hand mouse button. No attempts are made to do displays and changes to parameters will be lost.

The [DEFAULTS] option resets the default parameters in the VECTOR menu. This provides a quick method of returning to sensible values after experimentation.

The [DRAW] option takes the VECTOR settings and calls the vector display routine. In displays with more than one window ***FLOWVIS*** will prompt for the target window in which to display VECTORS. The vectors will be displayed for the currently selected data region using the active view orientation.

The vector head open [>] and display [>] options allow vectors to be drawn with closed heads or without arrow heads respectively. The appearance of the vectors is shown in the box to the left-hand side of the menu.

The colours in range [<][>] and [Min][Max] options allow the use of a limited number of colours rather than the whole range of available colours. The default is to use the full range of colours.

The colour range high limit [<][>] and low limit [<][>] options allow the range of colours used to be changed. The default is for the maximum allowed range of colours with red as the lowest valued and blue as the highest. The range can be reduced, reversed or swapped to white only (using the middle mouse button for selection).

The show vector [SYMBS] option allows the display of symbols to represent vectors which are nearly directly towards or away from the screen and therefore cannot be displayed as arrows. The symbols used are a cross for a vector tail (vector into the screen) and a circle for a head (vector coming out of the screen). The symbols are shown in the appearance graphics box if activated.

The vector set [-1] and [+1] options allow the selection of other orthogonal vector sets for display (if these are available in the current data-set). The component names for the indicated set are shown at the bottom of the menu for the X-, Y- and Z-components. By default the first set is always the first phase cartesian component set (if any vector sets are defined).

The colour [BACK] and [NEXT] options allow the type of vector colouring to be selected. The default is to use the resultant magnitude (size) for colouring but any variable from the data-base can be used.

The colour range and number of colours are used as defined in those options.

The magnitude range upper limit [<][>] and lower limit [<][>] options allow the vectors to be displayed for a limited range of magnitudes. The full range of vector magnitudes is the default and these values are displayed between the upper and lower limits.

The show vector [DOTS] option allows tiny vector components (that cannot be displayed as arrows) to be drawn as dots. These dots tend to clutter up the display if activated so this option has been provided to remove them.

The vector scale [LARGE] and [SMALL] options allow the displayed arrows to be scaled up or down in size. A value of 1.0 is the normal size based on averaged cell dimensions.

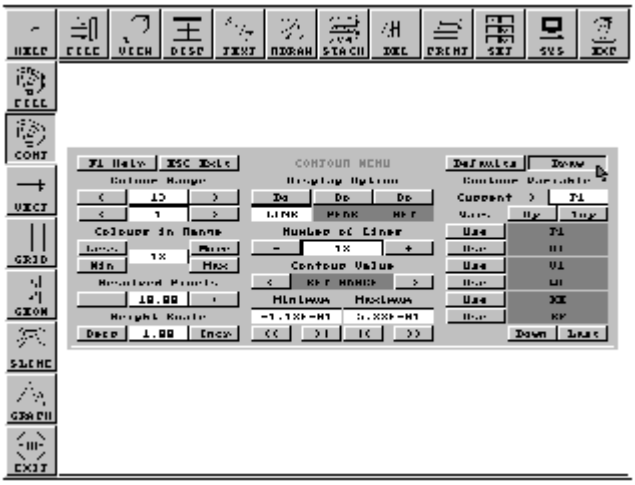
The vector head size [INCR] and [DECR] options allow the size of the vector arrow heads can be made larger or smaller depending on the value set for this option. The range is 0.1 to 0.9 times the overall vector arrow length.

The vector head angle [+] and [-] options allow the splay angle of the vectors to be set. The value is between angles of 5 and 80 degrees. The appearance of the vector arrows is visible in the vector appearance box on the left of the menu.

The X-, Y- and Z- component [+] and [-] options are not currently available. The display fields do show the current components for the indicated set. These options are being developed to allow any scalar variables to be treated as orthogonal vector components. These facilities will be available as an upgrade in the near future.

CONTOUR LINES

Contour lines overview



Contour lines use a single data-base variable to draw coloured iso-valued lines on a plane by plane basis for the current view region.

There are options for altering the colour range, the range of contour values and the number of lines that are displayed.

A single contour line is displayed using the "CONTOUR VALUE" option or a number of lines can be displayed equally spread over a "CONTOUR RANGE" of values.

By default the contours are displayed using one line for each of the available colours and the full range of values of the chosen variable.

The list of variables on the right hand side of the menu scrolls to allow access to any of the variables in the data-base. The current contouring variable is displayed just below the [DRAW] button.

The contour line display is the default but there are also options for a contour peak plot or a contour surface (net) plot. The contour peak plot uses the value of the variable being contoured to raise or lower the contour line from the plotting data-plane. This produces a 3-Dimensional surface when viewed from near edge on..

The "HEIGHT SCALE" option is used in conjunction with contour peak plots to manually adjust the height of the 3-Dimensional contour surface above the data-plane.

The contour net facility follows contours through three directions in 3-Dimensional space to form surfaces of equal contour value. The surfaces have the appearance of a net and hence the name contour nets. These contour nets cannot be displayed for 2-Dimensional data sets.

The resolution parameter termed "Resolved Pixels" determines how finely the contouring units are sub-divided so that the contour lines can be plotted. A small value will produce a more accurate display but will also take longer to draw.

Contour lines options

Use the [HELP] button to enter help mode for the CONTOUR-line sub-menu. The selection of any button in help mode will bring up help for that button or group of buttons. The help mode is exited by selecting on the [HELP] button again. The [F1] key will also perform the same action but will present help for the last highlighted button.

The [EXIT] option allows the CONTOUR-line sub-menu to be exited without any changes taking effect. This action can also be performed with the [ESC] key or the right-hand mouse button. No attempts are made to do displays and changes to parameters will be lost.

The [DEFAULTS] option resets the default parameters in the CONTOUR-line menu. This provides a quick method of returning to sensible values after experimentation.

The [DRAW] option takes the settings and calls the appropriate contouring display routine. In displays with more than one window

FLOWVIS will prompt for the target window in which to display contour-lines. The contours will be displayed for the currently selected data region using the active view orientation.

The contour colour range upper limit [<][>] and lower limit [<][>] options allow the colour range used for contouring to be selected.

The integer numbers displayed represent colours given that the lowest number represents red and the highest is blue. The range can be reduced, reversed or swapped to white only.

The [DO] contour LINE option is the default option for this menu. When this option is selected the contour plot will be of normal contour lines. The [DRAW] option must be used to start drawing the contours. The variable used for contouring is that indicated by the current contour variable text in the top right corner of the contour line menu.

The [DO] contour PEAK plots are available from this menu. These are formed by projecting the contours away from the data-plane by an amount which is proportionate to the value of the contour line value. These plots only have significance if viewed nearly edge on. The height scale option allows the height of the peaks to be changed.

The [DO] contour NET displays allows contour surface plots to be created. These can only be displayed for 3-Dimensional data-sets.

The surfaces are displayed throughout the domain. Again the contour variable is that indicated in the top right corner of the menu.

The contour variable [USE] options allow a named variable to be selected from a scrolling list. These options make the adjacent variables current and the current variable is used for the contour plot when [DRAW] is selected.

The scroll buttons [UP], [DOWN], [TOP] and [LAST] can be used to access more variables than can be displayed on the visible portion of the list. All of the variables from the data-base are presented in the list for contouring.

The number of contour lines [-] and [+] options are used to select the actual number of lines used for the contour plots. This can be larger than the number of colours available but this will produce plots that have a number of contours of the same colour.

The colours in range [LESS], [MORE], [MIN] and [MAX] options are used to select the number of colours to be used from the colour range. The default is the same as the number of colours in the colour range. This option is really only provided for using less than the maximum number of colours but still taken from the full range of colours.

The contour value [<] and [>] options allow a single contour value to be selected for display by the contour routine. The default and alternative to this option is to use a range of values.

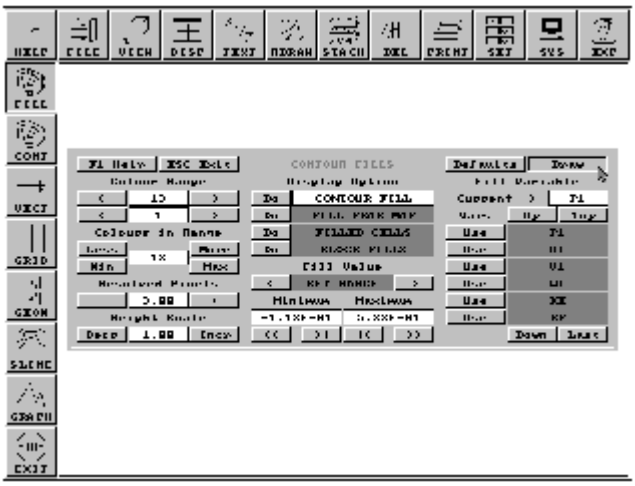
The resolved pixels [-] and [+] options allow the refinement of the contour display to be specified. A low value means that cells are sub-divided for more accurate contours. Accuracy does imply a slower display. The numbers actually represent the number of screen pixels that are used as a test for suitable refinement.

The height scale [DECR] and [INCR] options set the scale value for contour peaks. This parameter is used to adjust the height of the peaks from the plane. The scaling is automatic based on averaged dimensions but this option provides an override.

The minimum [<<][>|] and maximum [|<][>>] options allow the range of contour values to be limited. By default this is set to the full range for the currently selected variable but a sub-range of values can be selected. If a sub-range is chosen then contours outside of this range are not displayed.

CONTOUR FILLS

Contour fills overview



Contour fills draw solid filled contours for a chosen variable. Filled contours are also drawn on a plane by plane basis. The filled contours are actually ranges of contour values (bands) which are all coloured similarly within each range. The ranges are determined by sub-dividing the full range of values of the selected variable by the number of contours to be used in the display.

The choice of colours used is specified by the colour range and colours in range options. The colour range determines which colour represents high values and which represents low values given that colour number 1 is red and colour number 13 (at least on 16 colour displays) is blue. Three colours are reserved for the background, foreground and grey colours. The "COLOURS IN RANGE" option allows a sub-set of colours to be used from the full range.

Other than the default option for contour fills the FILL MENU also has options for fill peak maps, solid filled cells and block filled cells. The fill peak maps are analogous to the contour line peak plots and display a 3-Dimensional surface by raising the pixels being filled away from the data-plane by an amount which is proportional to the value of the variable at that point. The height scale option can be used to alter the height of the 3-Dimensional peaks from the data plane.

Contour fills overview

The contour filled peak plot should only really be used for a single data plane and for best results this plane should be viewed from near edge on. The filled cells option takes the field value within a cell and fills the whole cell according to that value. This can be useful for data analysis because the value displayed will be the same as that used in the numerical simulation code that produced the results. The block filled cells again uses the field value of a variable in a cell but draws a 3-Dimensional brick in an appropriate colour. The height of the block is proportional to the value of the variable. This option gives the appearance of a 3-Dimensional histogram for a plane of data.

Use the [HELP] button to enter help mode for the contour-FILL sub-menu. The selection of any button in help mode will bring up help for that button or group of buttons. The help mode is exited by selecting on the [HELP] button again. The [F1] key will also perform the same action but will present help for the last highlighted button.

The [EXIT] option allows the contour-FILL sub-menu to be exited without any changes taking effect. This action can also be performed with the [ESC] key or the right-hand mouse button. No attempts are made to do displays and changes to parameters will be lost.

The [DEFAULTS] option resets the default parameters in the contour-FILL menu. This provides a quick method of returning to sensible values after experimentation.

The [DRAW] option takes the settings and calls the appropriate contour fill display routine. In displays with more than one window ***FLOWVIS*** will prompt for the target window in which to display fills. The fills will be displayed for the currently selected data region using the active view orientation.

The colour range upper limit [<][>] and lower limit [<][>] allow the range of colours used in the fill displays to be selected. The range can be reduced or reversed for contour fills. The low valued colour is red and the high value is blue.

The [DO] CONTOUR FILL option is the default for this menu. The contour fill uses the current variable and settings to produce solid filled contour bands for the current data region. The contour fill display is initiated by selecting on the [DRAW] button.

The [DO] FILL PEAK MAP option allows a 3-Dimensional raised surface plot to be created. The height of the surface above the plane is proportionate to the local value being contour filled hence producing a terrain type plot with mountains and valleys. These only have significance if viewed nearly edge on. The height scale allows the height of the peaks to be changed.

The [DO] FILLED CELLS option produces a plot with solid filled cells where each cell has one colour appropriate to the selected variable field value within that cell.

The [DO] BLOCK FILLS option uses a single data plane to draw hidden line blocks of appropriately scaled height and colour. These are most effective when viewed from near edge on to the plane as they produce a 3-Dimensional histogram of the data values on a data plane. The height scale options can be used to scale the height of the blocks.

The fill variable [USE] options allow a named variable to be selected from a scrolling list. These options make the adjacent variables current and the current variable is used for the fill plot when [DRAW] is selected.

The scroll buttons [UP], [DOWN], [TOP] and [LAST] can be used to access more variables than can be displayed on the visible portion of the list. All of the variables from the data-base are presented in the list for contour filling.

The colours in range [LESS], [MORE], [MIN] and [MAX] options are used to select the number of colours to be used from the colour range. The default is the same as the number of colours in the colour range. This option is really only provided for using less than the maximum number of colours but still taken from the full range of colours.

The fill value [<] and [>] options allow a single contour fill value to be selected for display by the contour filling routines. The default and alternative to this option is to use a range of values.

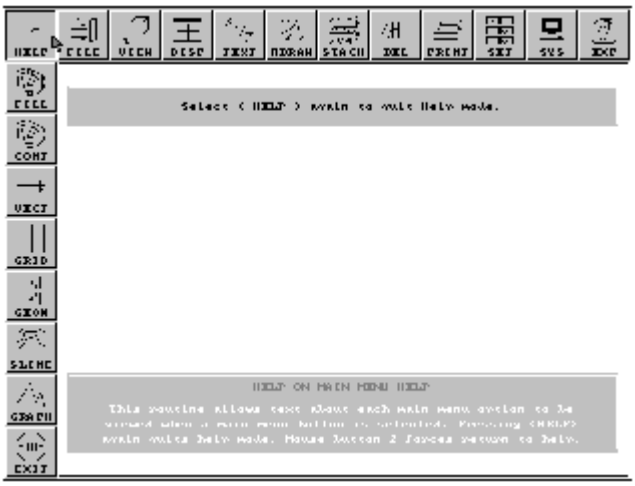
The resolved pixels [-] and [+] options allow the refinement of the contour fill display to be specified. A low value means that cells are sub-divided for more accurate filled contours. Accuracy does imply a slower display. This actually represents the number of screen pixels that are used as a test for suitable refinement.

The height scale [DECR] and [INCR] options set the scale value for contour fill peaks and block fill plots. This parameter is used to adjust the height of the peaks from the plane. The scaling is automatic based on averaged dimensions but this option provides an override.

The minimum [<<][>|] and maximum [|<][>>] options allow the range of contour fill values to be limited. By default this is set to the full range for the currently selected variable but a sub-range of values can be selected. If a sub-range is chosen then contour fills outside of this range are displayed as for the extreme limits of the fill colour bands.

HELP

Help overview

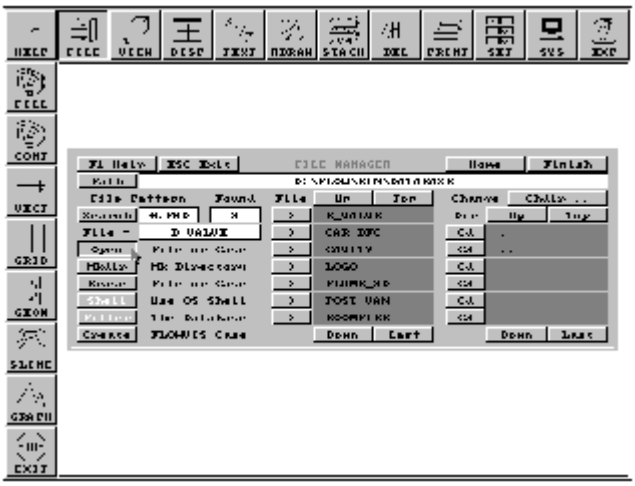


FLOWVIS has an on-line help system that is accessible from the main menu and pop-up sub-menus. Help mode is entered by selecting on the [HELP] option or by pressing the [F1] key. The help system will then display a message in the top half of the screen to inform you that the system is now in help mode and an information box towards the bottom of the screen that contains the help text. (The format for displaying sub-menu help is identical). Once help mode has been entered the system waits for you to select on an option. Selecting any valid option will bring up the on-line help associated with that button. Finally the help mode is exited by again selecting on the [HELP] option or pressing [F1].

The main menu help system gives general help about the main menu options whereas the sub-menu help gives specific help about options or groups of options within individual menus.

FILE MANAGER

File manager overview



The FILE MANAGER menu has facilities for opening various types of file, moving through the hard-disk directory structure, making directories, erasing individual files and data-sets and for creating ***FLOWVIS*** data sets from input files containing field data.

Usually the FILE MANAGER is used to load a data set. The files used by ***FLOWVIS*** are all named by an 8 character case name. This identifies all of the files of the particular case. ***FLOWVIS*** uses the 3 character extension to determine the purpose of individual case files. You should not rename ***FLOWVIS*** case files individually but you can rename all of a case. e.g. you may rename a data-set called "OLDCASE" to "NEWCASE" using the DOS command:-

RENAME OLDCASE.f* NEWCASE.f*

FLOWVIS will then be able to connect "NEWCASE" for analysis and visualisation.

FLOWVIS *only has one data set attached at a time but keeps the case names in the redraw stack for seamless visualisation of more than one data-set. You should not erase data sets for which there are commands in the redraw stack as* ***FLOWVIS*** *will not be able to redraw these commands.*

When [OPEN] is selected, for a data-base case, any previously open data-base files are closed and the new ones opened. The [SEARCH] option is provided so that ***FLOWVIS*** is aware of the type of file you are trying to open. The search pattern specifications are:-

"*.DAT"

This is for input data files that need to be converted to the ***FLOWVIS*** format. This is only for preference. ***FLOWVIS*** will actually create a data-case from any named file provided it is a field data file.

"*.FHD"

This stands for ***FLOWVIS*** HeaDer files. These contain the general information about the data-cases that inform ***FLOWVIS*** about the number and type of field variables stored and the extent and size of the data domain.

".DSC"*

These files contain saved stack information. Loading a stack file in the file manager will cause the commands in that stack to be replayed. The data-sets used in a stack must still exist in the same directory location for ***FLOWVIS*** *to be able to replay the commands.*

"*.*"

This search pattern displays all of the files in the current directory and can be used for browsing. There is currently a limitation of 250 file name entries in the current listing.

The current directory path is displayed at the top of the menu next to the [PATH] option. ***FLOWVIS*** requires that you use the [PATH] option to change drives on DOS systems. This is necessary for maintaining compatibility with unix-like systems where there is no such concept as a drive. You may also use [PATH] to type in a known path without having to use multiple [CD] selections to navigate the directory structure. If a full typed path is incorrect then ***FLOWVIS*** leaves the path that was previously set. This is also true of disk drives that do not exist or empty floppy drives.

If you get lost in a directory structure then the [HOME] option can be used to return you to the directory that ***FLOWVIS*** was initially run from.

The FILE MANAGER is also called by several other menus for getting an existing file name or for just opening a new data-set if required. When the FILE MANAGER is used in this mode it will not allow you to change the search specification and the menu itself will only pass back the file-name and path requested by the calling menu when [OPEN] or [DONE] are selected.

See the manual section about "CREATING A ***FLOWVIS*** DATA-CASE" for details of preparing field data files for visualisation by ***FLOWVIS***.

Opening a file is achieved by selecting on the [>] button next to the required file. This name will then become visible in the FILE = <filename> display field and the cursor moves to highlight the [OPEN] button. Selecting on the [OPEN] button will then open that file and close the FILE MANAGER menu.

File manager options

Use the [HELP] button to enter help mode for the FILE MANAGER sub-menu. The selection of any button in help mode will bring up help for that button or group of buttons. The help mode is exited by selecting on the [HELP] button again. The [F1] key will also perform the same action but will present help for the last highlighted button.

The [EXIT] option allows the FILE MANAGER sub-menu to be exited without any changes taking effect. This action can also be performed with the [ESC] key or the right-hand mouse button.

The [HOME] option sets the current directory path to the start-up directory path regardless of the current path or drive. This is useful if you get lost in an obscure directory.

The [FINISH] is use to exit from the FILE MANAGER. No data is lost and any attached files remain so. A file must be [OPEN]ed for viewing before any of the other main menu display commands become active.

The [PATH] option can be used to jump to a known directory path. This must be typed in full using the text input box. The alternative is the scrolling list of directories on the right. The [PATH] option must be used to change disk drives on DOS based systems. If the typed path does not exist then the previous path will be restored.

The [CHDIR..] option allows the current directory to be set to the parent directory. This is a way of navigating back up a directory structure. See also [Home] to get to the home directory.

The file and directory [UP], [DOWN], [TOP] and [BOTTOM] options are used to scroll the file and directory name lists. The file and directory lists maintain file and directory names for the current path. These lists can be scrolled (if required) by using the scroll buttons provided. The selection buttons for the lists are provided on the left.

The file pattern [SEARCH] option allows the file search extension to be changed. A formatted data-base is "*.FHD", all files are "*.*", stack files are "*.DSC" and unformatted input files are "*.DAT". When ***FLOWVIS*** is requested to search for ***FLOWVIS*** data-cases only the 8 character identifying name is displayed in the file name list.

The file list [>] options allow the adjacent filename to be made current. The current file is shown by the FILE=<filename> message.

This filename is used for the options in the bottom left of the FILE MANAGER menu.

The directory list [CD] options perform a change of directory to the named directory immediately below the current path. The [Home] option can be used to return to the start-up directory if the user becomes lost.

The [OPEN] file or case option uses the current file to try to load a file. If the FILE MANAGER is being used for data-base access then all files for that case are opened and all other files closed. If the search pattern is set for a previously saved stack file then the stack file is loaded and replayed. The FILE MANAGER can also be used by other menus to return a file name and a directory path. These are returned when the [OPEN] or [FINISH] buttons are selected.

The [MKDIR] option allows the creation of a named sub-directory below the current path or at a fully specified path. The name must be typed in the text input box.

[ERASE] allows a single file to be deleted or an entire case with all associated files. The Shell tool will provide an alternative means of executing OS commands when available.

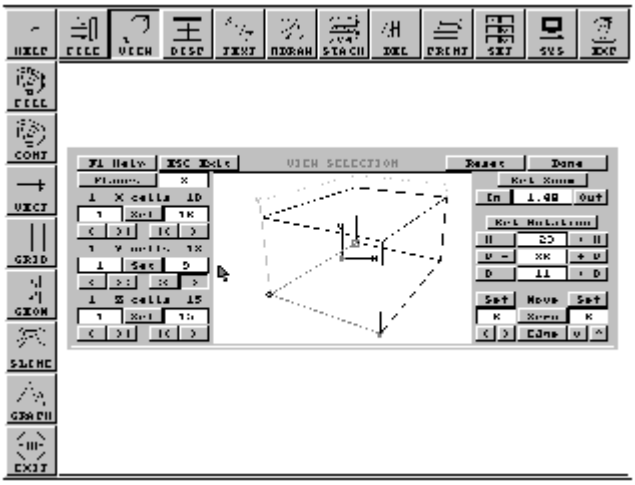
The [SHELL] operating system shell option is currently under development. When available it will provide facilities for file copying, deletion and renaming, etc. This will be provided as an upgrade.

The [FILTER] facility is currently under development. When available it will provide access to data-base manipulation routines. These capabilities will be provided as an add on enhancement to ***FLOWVIS***.

The [CREATE] a ***FLOWVIS*** data-case option manipulates an input data file to produce a ***FLOWVIS*** data-set. The data-base can be created from this option or from the command line utility provided with ***FLOWVIS***. After creating a data-set ***FLOWVIS*** will ask if you now want to load the data-set for visualisation and analysis.

VIEW SELECTION

View selection overview



The VIEW SELECTION menu provides facilities to set limits for the data region to be used for subsequent displays and to set the view orientation that any data is displayed from.

The view menu consists of three vertical sections. The left most section allows the data extent to be limited and the plane type to be selected. The selection of a data region is indicated by the cell indices in the three orthogonal data-base directions X, Y and Z. The extent of the data to be visualised by subsequent commands is then indicated by three pairs of low and high cell index limits. A visual feedback of the selected region is visible in the graphical area as the solid-line boundary.

The cell limits can also be selected by coordinate values. These are the [SET] buttons near the cell limit indicators. Using set will present a text input bar which will display the current coordinate range for the chosen direction. Once you have entered the required range the coordinates will be mapped to the nearest cell indices.

The middle section of the view menu contains a graphical representation of the data domain and the selected data region. Also present are the data-base coordinate axes, the view rotation axes (labelled H, V and D) and if the data-set contains volume porosity data there will also be points indicated to show the location of blocked cells.

The domain outline is shown dotted and the selected region is shown as a solid-line boundary. Generally speaking the image in the graphics area will change interactively as the view menu options and parameters are changed. Also this image represents the images that will be produced in the display area windows. This is not strictly true of target display windows which have a very different aspect ratio from the view menu graphics area. In these cases the height of the image in the target window is approximately proportionate to the height of the image in the view menu. This means that short, wide target windows will show an image which is well in from the left and right borders if they were touching in the view menu and that tall, thin target windows will generally loose the left and right edges of the image. It is recommended that the display type be selected so that the windows match the shape of the domain as closely as possible.

Finally the right most side of the view menu has the options for view orientation manipulation. These include zoom-in, zoom-out, rotation (about three orthogonal directions) using the indicated --> H, --> V and --> D axes and finally image positioning by stepped shifting in a vertical or horizontal direction. Again these options have immediate visual confirmation from the graphical area.

These options should be used to orient the data on screen as you require it to be drawn by subsequent display commands. If you loose sight of the domain or get an obscure view then the

[RESET] option can be used to reset the default view. Selecting [Done] will then return to the main menu ready for drawing (or redrawing) commands using the newest view settings.

The view menu is also seen in three slightly different forms for getting region, slab and line limits respectively. In these data selection menus the view orientation options still work within the menu but are not used to update the current orientation settings. These menus are used to visually select regions for use by the GRAPH and STREAMLINE options.

View selection options

Use the [HELP] button to enter help mode for the VIEW SELECTION sub-menu. The selection of any button in help mode will bring up help for that button or group of buttons. The help mode is exited by selecting on the [HELP] button again. The [F1] key will also perform the same action but will present help for the last highlighted button.

The [EXIT] option allows the VIEW SELECTION sub-menu to be exited without any changes taking effect. This action can also be performed with the [ESC] key or the right-hand mouse button.

The [RESET] option will reset all of the view orientation settings to the default values. This can be invaluable when the domain has been moved of the screen or zoomed in too far.

The [DONE] option implements all of the changes made in the menu and exits from the menu. This contrasts with [EXIT] which discards any changes made. Control returns to the main menu ready for visualisation commands. The view orientation settings are always compared with the view selection of windows with existing commands. ***FLOWVIS*** will not permit two different views to be mixed in a window so you will be prompted (as you try to draw or redraw) if the views have changed. The choice presented is to use the existing view or to update the window to use the new view.

The [PLANES] option allows the orientation of data to be selected. Since most of the display commands are drawn plane by plane this will determine how the displays are created. The View menu has three data selection forms that only allow selection of a region, slab or line of cells. These are used by other menus that need to allow a region to be selected visually. The equivalent buttons in the three data selection menus are [SLABS], [SLAB] and [LINE].

The zoom setting [SET ZOOM], [IN] and [OUT] options allow the level of zoom to be set explicitly or incrementally. The display field shows the level of zoom as a real number. A zoom value of 1.0 is normal size whilst -ve values zoom out and +ve zoom in. A factor of 2.0 for example makes the object twice the size. The [SET ZOOM] option uses a text-input box to prompt for a real value. The zoom will change the appearance of the view seen in the graphical area of the menu. Use zoom to bring the domain closer or to zoom into a specific area.

The X-cells range lower limit [<][>|] and upper limit [|<][>] options are provided to allow the selection of the range of cells from which data will be viewed in the X coordinate direction (in Cylindrical Polar coordinates this is the angle direction). These options change the cell indices incrementally.

The X-cells range [SET] option uses a text input box for the entry of two coordinate X values which are converted to the nearest cell indices. When called the current coordinate range is displayed. The full range of cells in this coordinate direction is indicated just above the button.

The move [SET], [<] and [>] options allow for shifting the display horizontally. The value displayed is a percentage of the maximum data-base dimension. This value can be set explicitly or interactively using the two buttons below the indicator. The effect of shift horizontally is to move the image within the display area. A value of 0 means that the display is centred horizontally.

The move [SET], [v] and [^] options allow for shifting the display vertically. The value displayed is a percentage of the maximum data-base dimension. This value can be set explicitly or interactively using the two buttons below the indicator. A value of 0 means that the display is centred vertically.

The [ZERO] option centres the data-base in the centre of the screen. This is used for bringing the image back into view when it has been shifted too far from the centre. The effect is analogous to setting both the horizontal and vertical shift values to zero.

The move to [EDGE] option is currently under development. This will shift and zoom the data-base for the maximum possible screen usage. This feature will be available as an update.

The [SET ROTATION] option allows all three axial rotation angles to be set explicitly in a text input box. The ordering is H-axis, V-axis then D-axis rotation in that order. The rotations will be applied in that order. Since rotations are non-commutative it is critical that the order be observed. This option is provided to repeat known rotations as displayed by the rotation angles given below this option. The alternative is to use the interactive mode which is more intuitive because the rotations are applied to the graphical image area incrementally.

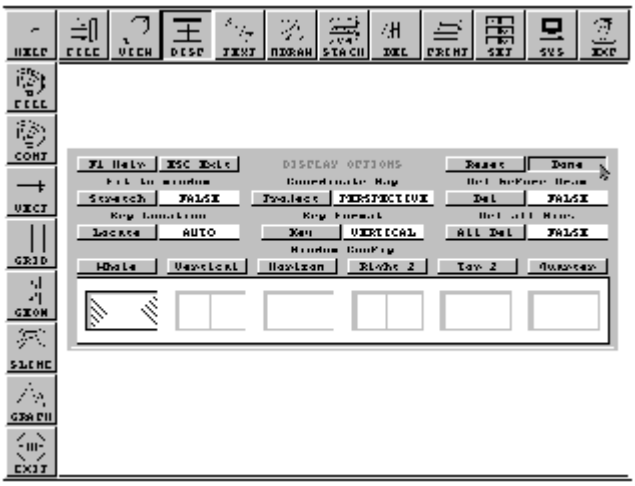
The Y-cells range [SET], lower limit [<][>|] and upper limit [|<][>] options are provided to allow the selection of the range of cells from which data will be viewed in the Y coordinate direction (in Cylindrical Polar coordinates this is the radial direction). The [SET] option uses a text input box for the entry of two coordinate Y values which are converted to the nearest cell indices. When called the current coordinate range is displayed. The other options change the cell indices incrementally. The full range of cells in this coordinate direction is indicated just above the buttons.

The rotation [H-], [H+], [V-], [V+], [D-] and [D+] options allow incremental rotation about the screen-(H)orizontal, screen-(V)ertical and screen-(D)epth axes respectively. The rotations are applied about either global or local orthogonal axes. The global axes remain fixed in space whereas the local axes follow the domain as it is rotated. The angles for each axial rotation represent the angles that would need to be applied in a (H, V, D) order to get to the same resultant image. This is necessary for the [SET ROTATION] option which allows all three axial rotations to be set explicitly. The angle step size is set in the SETTINGS MENU but is by default 5 degrees. Because of the nature of rotation the numerical angles displayed do not necessarily change predictably BUT the object does rotate predictably which is somewhat more important.

The Z-cells range [SET], lower limit [<][>|] and upper limit [|<][>] options are provided to allow the selection of the range of cells from which data will be viewed in the Z coordinate direction (in Cylindrical Polar coordinates this is the length direction). The [SET] option uses a text input box for the entry of two Z coordinate values which are converted to the nearest cell indices. When called the current coordinate range is displayed. The other options change the cell indices incrementally. The full range of cells in this coordinate direction is indicated just above the buttons.

DISPLAY SELECTION

Display selection overview



The DISPLAY MENU provides various facilities for configuring the display. The three main features are the type of coordinate mapping, the colour key settings and the number and type of windows used in the main display area.

The coordinate mapping can be either parallel projection or perspective projection. By default the perspective projection is used for 3-Dimensional domains (because it gives a more realistic appearance) and parallel is used for 2-Dimensional domains.

The key legends for colour displays are boxes that indicate the values associated with colours for certain commands. For example a vector display can have a key that indicates that vectors coloured in red have a certain magnitude whilst vectors coloured in blue have a different magnitude. There are options for how the key is to be placed and the type of key to be drawn. The placement can be "automatic" in which case ***FLOWVIS*** *places the keys or "manual" in which case the key is placed as an outline by using the mouse.*

There is a third option which makes ***FLOWVIS*** *ask how the key should be placed every time. Currently there is only a choice of drawing a vertical key or not having any keys at all. The display area can have up to four separate windows. The options for the type of windows are available at the bottom of the DISPLAY MENU. The choices of windows range from one window which uses the whole of the display area to four quarter-windows. The current selection is indicated by a pictorial representation of the main display area, in the graphics box at the bottom of the menu.*

Display selection options

Use the [HELP] button to enter help mode for the DISPLAY sub-menu. The selection of any button in help mode will bring up help for that button or group of buttons. The help mode is exited by selecting on the [HELP] button again. The [F1] key will also perform the same action but will present help for the last highlighted button.

The [EXIT] option allows the DISPLAY sub-menu to be exited without any changes taking effect. This action can also be performed with the [ESC] key or the right-hand mouse button.

The [RESET] option resets the parameters in the DISPLAY menu back to the default settings.

The [DONE] option performs an exit from the DISPLAY menu but any changes made are left intact. Control passes back to the main menu. If the window configuration has changed then ***FLOWVIS*** will re-display the main graphics area with the new windows. If there are commands

in the stack then ***FLOWVIS*** will ask if you wish to proceed with the re-configuration. Sometimes keys (which have absolute screen pop-up locations) do not go well in other windows.

The fit to window [STRETCH] option allows the selected data region to be stretched out to fill the whole screen. This does not preserve the aspect ratio of the domain but can be useful for visualising long, thin domains.

The type of coordinate mapping is specified by the [PROJECT] option. ***FLOWVIS*** supports both parallel and perspective projections. Perspective gives 3-Dimensional domains a more realistic look on the screen but 2-Dimensional domains use parallel projection by default.

The delete before draw [DEL] option allows ***FLOWVIS*** to clear the screen before every drawing command is executed. By default ***FLOWVIS*** places images one on top of another, where possible. This option allows each display to be drawn on a clear screen. All commands are still be saved in the stack for redrawing.

The key location [LOCATE] option informs ***FLOWVIS*** how to place the colour keys for coloured vector, streamline and contour displays.

Options are AUTO-matic, ASK as required and MANUAL which allows interactive key placement. The manual mode means that the key bounding box is displayed under user control and can be positioned interactively on the display. ***FLOWVIS*** uses pop-up keys that are removed when a new command is targeted on a window and replaced after the command has been drawn.

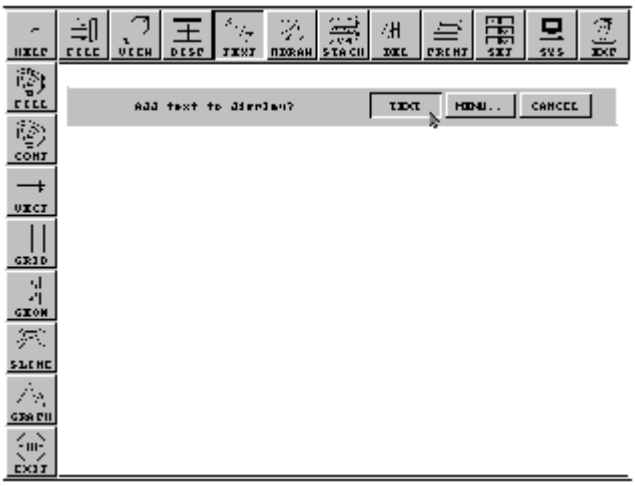
The key format [KEY] option only allows keys to be drawn vertically or not drawn at all. The keys contain numerical and colour mapping information for contour, vector and streamline plots. It is intended that horizontal keys will be provided in future versions of ***FLOWVIS***.

The delete all windows [ALL DEL] option configures how the main menu {DEL] option works. When this option is set a single selection if delete will clear all windows and remove the last redraw command from each window (for multi-window displays). This option has no effect for a single window display.

The [WHOLE], [VERTICAL], [HORIZON], [RIGHT 2], [TOP 2] and [QUARTER] options determine how the main display area is divided into windows. Each of the configured windows can have individual view orientations. These options provide a variety of window groupings as shown in the graphics box. When multiple windows are used for display ***FLOWVIS*** will ask you to select in a window to indicate where commands are to be displayed.

TEXT

Text overview



The text option allows displays to be annotated with text and, optionally, to have a pointer arrow.

The text to be displayed is typed into an input box and when [ENTER] is selected a rectangular box appears in the target display window.

This box is the outline of the text to be displayed. Selecting a location and pressing the left mouse button will result in the placement of the text at that position. Then ***FLOWVIS*** will ask if you require a text pointer. If you require a pointer this is placed interactively by selecting a point for the tail and then selecting a point for the head. Depending on the configuration of ***FLOWVIS*** an arrowhead can optionally be drawn on the pointer.

There will shortly be available a menu for the selection of font and text colour. This will be available as an upgrade.

REDRAW

Redraw overview

The redraw command takes all the commands for a window and replays them after first clearing the window. The redraw ordering can be by the drawing order or by a logical order that was set to prevent certain commands from obscuring other commands. The choice of the ordering is set within the SETTINGS MENU.

Redraw can also be used to redraw more than one window if the Redraw Globally option has been set in the settings menu.

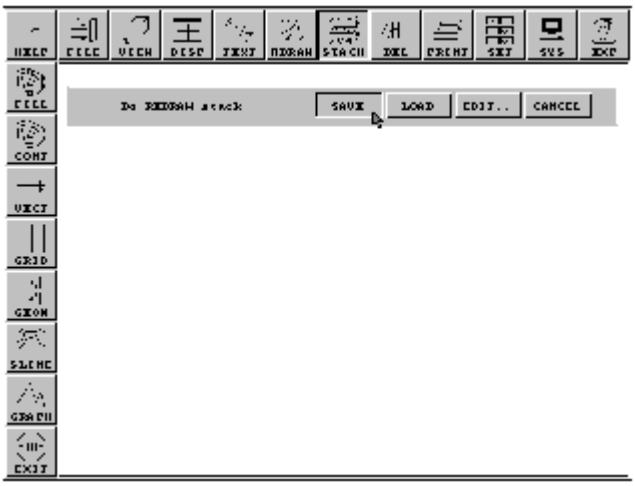
When a stack file has been loaded the redraw option can be used to replay the commands saved there.

By default the main menu [DEL]ete option clears the window and removes the last command from the redraw stack for that window. If there were many commands in a window then doing a redraw will re-display all the commands left for that window.

One further feature of REDRAW is that if the view orientation has changed from that of the commands already in a window and redraw is requested then ***FLOWVIS*** *will ask if the commands should be redrawn using the previous or the newest orientation. See also the notes on the VIEW MENU operation.*

STACK

Stack overview



The redraw stack is the repository for all successful drawing commands. The stack can be thought of as a number of slots into which commands and all their associated parameters and attributes can be saved. There are also a number of slots, which store window specific information so that views do not become muddled (at least that is the intention!). The stack actually resides in a file like structure that can be physically saved to a permanent file for subsequent reuse by ***FLOWVIS***. Since this file only contains the commands (but no data) the data sets that originally created the displays must still exist for successful replay of the stack. Furthermore the data sets must exist in the same locations because ***FLOWVIS*** keeps track of absolute paths to the data sets.

The stack file cannot be edited by hand as the commands and parameters are stored in binary access format.

Work is progressing on an editing facility which will allow commands to be pulled back into their menus for modification. This facility will be available with upgrades of ***FLOWVIS****.*

DELETE

Delete overview

The main menu [DEL]ete command has several different configurations but in its most basic form it clears a target window and removes the last added command (for that window) from the redraw stack.

When the display is set to one window then delete will bring up the previous picture automatically (as if the last command had not be selected).

Delete can be used to clear all windows and optionally to remove all commands from the redraw stack. See the DISPLAY MENU and the SETTINGS MENU information for more details.

In displays with more than one window delete will prompt you for a target window in which to delete.

A message will be posted onto the screen when the redraw stack for any window has been emptied.

PRINT

Print overview

FLOWVIS can produce hard-copy output files in a number of formats but does not print directly to printers. This is necessary because of the vast range of printers, printer drivers and languages for graphics printers.

FLOWVIS can produce two distinct types of image. The first is bitmapped images that are essentially screen dumps in various formats. These have a set resolution that is dependent on the screen resolution from which they were produced. The second type are vector graphic image formats that store a list of object definitions to re-create the final image. These are not limited to the screen resolution and generally produce much better hard-copy results on suitable printers. Vector graphic formats generally do require performance printers and can be very large files.

Currently the printer file-creation drivers in ***FLOWVIS*** *have to redraw the whole display whilst at the same time creating the required hard-copy file. Since most users will not require the main menu to be printed the print option will redraw on the full screen as it creates the hard-copy output.* ***FLOWVIS*** *will then wait for user input from the mouse or keyboard. This allows video or photographic images to be captured if required.*

The [OK] prompt box that appears at the bottom right hand corner of the display means that ***FLOWVIS*** is waiting for interaction after having created any required hard-copy output. The middle mouse button can be used to hide the [OK] box if necessary.

The image formats supported are HP-GL, PostScript and Raster. There is also a PCX screen grabbing utility built into ***FLOWVIS*** that can be activated by pressing [ALT] and [F2] together. This is almost always available except when text input is occurring to a text box.

The PCX grab utility will capture the whole screen including any menus or message text that may be present. A single speaker tone means that PCX file creation has started and two tones indicates that the PCX file has been created.

Generally all hard-copy files are saved in the current directory and have filenames made up of three parts. The first part is an identifier of 4 characters that you may specify for recognising the plot files. The second part is an incremented number (starting at 0001) that ***FLOWVIS*** *adds to make sure that any existing plots are not overwritten. Finally the extension indicates the type of hard-copy file. The extension ".HPG" is for HP-GL files, the ".PS" extension indicates postscript files, the extension ".RAS" indicates Raster files and ".PCX" extensions are for PCX bitmapped images.*

Print options

Use the [HELP] button to enter help mode for the PRINT sub-menu. The selection of any button in help mode will bring up help for that button or group of buttons. The help mode is exited by selecting on the [HELP] button again. The [F1] key will also perform the same action but will present help for the last highlighted button.

The [EXIT] option allows the PRINT sub-menu to be exited without any changes taking effect. This action can also be performed with the [ESC] key or the right-hand mouse button. No hard-copy output or full-screen display will be created if this menu is exited.

The [DEFAULTS] options will be available to reset the default print settings.

The [PROCEED] option performs the required hard-copy print and display. This will generally involve redrawing all picture elements on the full screen. This is necessary for hard-copy file creation because of the nature of the print drivers. When the full screen has been created

FLOWVIS draws a small box marked [OK] in the bottom right of the screen and waits for some interaction via the mouse or keyboard. This facility is provided to allow video or photographic Image capture from screen. The [OK] box can be removed by pressing the middle mouse button.

The printer driver [USE] options allow specific hard-copy printer files to be created. The currently supported selection includes HP-GL, PostScript and Raster image output. These files will be created in the current working directory.

The hard-copy status [HARDCOPY] option informs ***FLOWVIS*** to create hard-copy output as files defined by the formats above. If output is disabled then a full screen image is created but no hard-copy files are produced.

SETTINGS MENU

Settings menu overview

The settings menu allows many of the default settings in ***FLOWVIS*** to be changed. Some of the most useful are:-

Domain and Region Outlines:

This turns on or off the automatic display of the boundaries of the whole domain and currently selected data region respectively.

Min and Max Scanning:

This indicates to ***FLOWVIS*** *how the colour range is to be scanned for the values in a given data region. The domain option means that minimum and maximum values will be those of the whole domain rather than just the selected data region. The plane option uses only those planes which pass through the data region for minimum and maximum scanning whilst the region option actually checks only those cells within the specified data-region to find the minimum and maximum. These options are necessary when you want to use the full range of colours (for a small region) even when the region may have only a small range of values within it.*

Redraw ordering:

As mentioned in the redraw option the stack commands can be drawn in a logical order to prevent (as far as possible) overwriting. The alternative is to draw the commands in the first drawn order.

Plane Steps:

This parameter tells ***FLOWVIS*** to skip planes (where possible) when creating displays. This has the effect of rarefying the data when there is too much to visualise comfortably and when the screen becomes cluttered. Values of zero means do not skip any p1anes. Streamlines and graphs ignore this parameter completely. Contour lines, fills and grids skip planes only and vectors skip cells in all three data-set directions.

There are also a number of parameters for controlling the default step sizes used in the VIEW SELECTION menu.

Settings menu options

Use the [HELP] button to enter help mode for the SETTINGS sub-menu. The selection of any button in help mode will bring up help for that button or group of buttons. The help mode is exited by selecting on the [HELP] button again. The [F1] key will also perform the same action but will present help for the last highlighted button.

The [DONE] option performs an exit from the SETTINGS menu and returns to the main menu to await further commands.

The show outlines [DOMAIN] and [REGION] options allow displays to be configured to draw the outlines of the whole domain and the currently selected data-region respectively. These appear as dotted grey and solid white bounding lines respectively.

The choose window [CHOOSE] option informs ***FLOWVIS*** to prompt for a target window in multiple windowed displays. This option can be made to keep the current window active without prompts for a choice of window. By default ***FLOWVIS*** always prompts for a target window when there are more than one windows.

The min and max scan [SCAN AS] option allows colour coded pictures to use different minimum and maximum values for scaling. The options are for whole DOMAIN, whole PLANES that cut the data region or as the current active REGION. By default the DOMAIN is used for scanning minimum and maximum values because colour scaling for sub regions will be consistent anywhere in the domain. The REGION option is used where features cannot be seen because of a large range of values in the whole domain but small ranges within a selected data region.

The rotation type [ROTATE] option allows the rotation axes to be LOCAL or GLOBAL. LOCAL axis rotation moves the rotation axes with the domain whereas global axes are fixed in space and the domain moves past. Changing this option while there are existing stack commands may have undesirable results in terms of the images produced.

The redraw ordering [AS] option allows the saved stack commands to be replayed in a LOGICAL or the actual DRAWING order. The logical order helps to prevent display corruption and image overwrite by re-ordering the commands.

The redraw globally [GLOBAL] option allows the main menu [REDRAW] command to operate on all windows of a multiple windowed display. The alternative is for redrawing in a chosen (or active) window only. In the case of multiple window displays ***FLOWVIS*** will prompt for the target window for redrawing.

The default zoom [<] and [>] options allow the default zoom value to be set. This will be the value set for the zoom factor whenever [RESET] is selected in the VIEW menu in the current session.

The redraw after printing [+REDRAW] option allows the graphics image to be restored along with the main menu after a full-screen print. Alternatively ***FLOWVIS*** will merely replace the main menu after clearing the screen. The commands in the redraw stack are unaffected.

The delete removes all commands [REMOVE] option allows delete to remove all of the previously saved commands from the redraw stack when delete is selected. Use this option with caution because deleted commands are NOT retrievable. The default usage only allows the last command to be removed.

The zoom step sizes [LESS] and [MORE] options allow the size of zoom steps in the VIEW menu can be set. A smaller value gives greater control but also takes longer to make large changes in the zoom factor. Note that the VIEW menu does provide accelerated stepping for incremental options by using the middle mouse button.

The plane steps [-] and [+] options inform ***FLOWVIS*** to skip over cells where possible. In most cases this will result in stepping over planes only but vector displays step over cells in all three index directions whilst streamlines cannot step over cells at all.

This facility can be useful for huge data sets where images can become cluttered by the amount of information displayed.

The show views similarly [SAME XYZ] option allows the view onto the data-base to be independent of the type of plane selected in the VIEW men. This contrasts with bringing the selected plane flat onto the screen by applying default rotations.

The angle step size [LOW] and [HIGH] options allow the view menu rotation steps size to be changed. By default the angle step size is 5 degrees but this may be increased or reduced here. The step size is the number of degrees added to (or subtracted from) the current rotation about one axis.

SYSTEM

System overview

This option is provided to allow future enhancement of the settings available with ***FLOWVIS***.

This functionality will mirror the control provided by ***FLOWVIS*** *initialisation files. These facilities will be available as with upgrades.*

EXPERT

Expert overview

The EXPERT MENU is similar to the main menu SYSTEM option in that it allows for future enhancement of ***FLOWVIS***. The functionality to be added here is of a more practical nature than the system settings options. The reason for including the menu now is that the remainder of the ***FLOWVIS*** user interface will not have to change significantly (even as planned extensions are added). The enhancements to change significantly (even as planned extensions are added). The enhancements already being developed are listed in the section "PLANNED EXTENSIONS TO ***FLOWVIS***".

Currently there are three facilities provided in the menu:-

1 The demonstrator shows some saved images that were captured in internal image format and saved to disk files.

2 The menu re-display option redraws the whole screen in full.

3 The slide utility allows internal screen images to be saved to or loaded from files. These images require an eight-character identifier to which **FLOWVIS** adds an extension, which indicates the screen mode that the images come from. You cannot load images into different screen modes, than they were produced from, as this can lead to unsatisfactory or corrupted displays.

The screen image demonstrations were produced by the screen image LOAD and SAVE utilities. You can replace the existing "SCREEN??.SM1" files with your own if you require demonstration facilities of your own data sets.

LIMITATIONS OF *FLOWVIS* (Version 3.10)

1 Only databases of size 70 cells by 70 cells by 70 cells by 50 variables are currently supported. There is also currently a restriction of 70 cells in any one direction. These limitations are being removed in future releases of ***FLOWVIS***. Please contact the distributors if you have special requirements.

2 The redraw stack can hold up to 100 commands. After 100 commands the redraw stack can be saved and the existing commands overwritten. The stack always fills up from the start so it is unlikely that this size limitation should be too critical.

3 Only 10 graphs can be displayed in any one window.

4 Changes to default values and settings within ***FLOWVIS*** cannot be saved for subsequent re-use. Future releases of ***FLOWVIS*** will allow modifications to be saved and loaded as required. This is currently a way of tailoring the system by modifying the initialisation files by hand.

5 Graphs and text can be mixed within a window. If the graph elements are modified after text has been added then the text will be lost. There is also a limitation that text can only be added to a window containing graphs since adding a graph to a window that only contains text will result in the deletion of the text.

6 The colour key legends used in colour vectors, contour lines, fills and streamlines can be placed manually. There is a potential problem if one key is placed on an existing key because the pop-up key area saved can become corrupted. It is the responsibility of the user to make sure that manually placed keys do not overlap.

KNOWN BUGS IN *FLOWVIS* (Version 3.10)

1 Streamlines do not yet use porosity values correctly. This can lead to situations where streamlines enter blocked regions by up to half a cell. Work is in progress to correctly stop the streamlines from entering blocked regions or passing through obstructions.

2 Streamlines suffer from accumulated integration errors. This is more noticeable for true recirculations where there is a tendency for the streams to spiral outwards rather than meeting the original stream after a circuit of the recirculation. One possible way to overcome this is to have a fine grid.

FLOWVIS FACILITIES (Version 3.10)

User Interface

- Menu driven with pop-up sub-menus.
- Mouse pointing and selection.
- Cursor-key navigation with space bar or return-key selection.
- Intelligent next option selection in certain menus to speed up interaction. This involves placing the cursor on the next sensible button ready for immediate selection.
- Meaningful pop-up messages and critical action confirmation requests.
- Automatic command saves for subsequent replay and possible save to file.
- Break-in allowed on display creation by [ESC] or mouse button number [2] or [3].

Display

- Up to four display areas (windows).
- Up to four data sets on one display with one allowed for each window.
- Pointing selection / cursor selection of active window for direction of output.
- Selective redraw for all or single windows.
- Choice of six-window configurations e.g. whole area, split horizontal or quarter etc.
- Intelligent checking of display to prevent (as far as possible) corrupted displays caused by inappropriate mixing of commands.
- Textual annotation of displays.
- Upgrade capabilities for 256 colour displays.
- Upgrade facilities for very high resolution display modes (this requires a suitable graphics card and monitor).

Visualisation Options

- Contour line displays.
- Contour fill displays.
- Peak plots for 3-D visualisation of a data plane for contour fills and lines.
- Filled cells for data analysis.
- Filled cell 3-Dimensional histograms for a plane of data.
- Contour line iso-surface nets through 3-Dimensional data-sets.
- Vector displays for a vector set of up to three orthogonal components.
- Grid displays for cell-faces, cell-centres or grid outlines.
- Geometry display (defined by blocked grid cells with zero porosity).
- User defined geometrical features formed from boxes, lines, arcs, etc.
- Streamlines of Cartesian vector data.
- Colour coded displays with associate colour keys that can be located manually.

- Graph displays for lines of cells.

Data Selection

- By region specified by orthogonal minimum and maximum cell index limits.
- By entered physical coordinates that are mapped to nearest cell indices.
- Has visual confirmation of data selection limits and shows the domain boundary.
- Draws slab-by-slab for 3-D data region in a selected slab-wise direction.
- Automatic warnings if selected planes are not flat onto screen.

View Selection

- What-You-See-Is-What-You-Get view confirmation (this is consistent for windows of similar aspect to the view-menu display area).
- Three axes for combined rotations using global or local rotation strategies.
- Zoom in or out.
- Parallel (cabinet) or perspective 3-Dimensional projection.
- Quick porosity geometry (if available) and local axes display for ease of orientation.
- Move display within a window (up, right, down, left or centred).

Grid Types

- 2-D or 3-D structured meshes only.
- Regular Cartesian, Cylindrical Polar or Body Fitted Coordinates.

User Preferences

- Many features are set in run-time loaded files for easy configuration. User defined preferences will also be provided which can then be saved.

Hard-copy Output to File

- Plot-file creation using redraw (uses incremental naming to prevent overwrites).
- Support for Postscript, HP-GL and Raster formats.
- PCX bit-mapped screen image saves.

Data-base Creation

- Support for most structured mesh CFD code data formats. (Contact the Distributor for more details).
- Internal and external database creation tools to prepare data for visualisation by the ***FLOWVIS*** system.
- Data is displayed in the same form as it is used in the numerical simulation.
- ***FLOWVIS*** presents data accurately for data analysis.
- A fast access database is created for interactive visualisation but without extensive data proliferation seen in many other visualisation systems.
- Data sets are easily identifiable by case name and file extensions.

- The filter source code can be made available for inclusion in third party CFD software for seamless usage of ***FLOWVIS*** and your own CFD code.
- Fast but comprehensive data preparations for quick image creation.
- Potentially huge data sets can be visualised because ***FLOWVIS*** does not attempt to read all of the data into memory at one time.

A Post Processing analysis tool specifically for Fluid Dynamics researchers

- ***FLOWVIS*** is developed by CFD researchers for CFD users.
- Interactive techniques give you a better feel for your data.
- The production of high quality hard-copy files save you time.
- The use of innovative display techniques presents your data in the most meaningful way possible. The capabilities are also being continually enhanced.

PLANNED EXTENSIONS TO *FLOWVIS*

Display

Redraw editing past previous command deletion.

Presentation facility for screen displays.

User Preferences

Allow selective saving and loading of user defined defaults at runtime.

Access to low level colour palette.

Visualisation Options

Enhanced Graph facilities for annotation and presentation display style.

Particle tracking capability.

Display animation.

Operating System

Operating system interface for the target hardware.

- Porting to workstation hardware platforms.

Utilities

- Provision of data manipulation and preparation routines for input data files.

Research is being conducted into providing a limited capability CFD solver, with problem specification routines, available within the ***FLOWVIS*** environment with suitable monitoring capabilities. Whilst this cannot hope to be a competitor for commercial CFD solvers it will be suitable for teaching purposes.